NAVAL POSTGRADUATE SCHOOL

MONTEREY, CALIFORNIA

THESIS

MODELING AND SIMULATION OF SURVIVABLE ARMOR DESIGN STUDIES FOR IED THREATS

by

Douglas L. Kaminski

March 2008

Thesis Advisor:	Ramesh Kolar
Second Reader:	Garth Hobson

Approved for public release; distribution is unlimited

THIS PAGE INTENTIONALLY LEFT BLANK

REPORT DOCUMENTATION PAGE			*Form Approved OMB No. 0704-0188*
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instruction, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188) Washington DC 20503.			
1. AGENCY USE ONLY *(Leave blank)*	**2. REPORT DATE** March 2008	**3. REPORT TYPE AND DATES COVERED** Master's Thesis	
4. TITLE AND SUBTITLE Modeling and Simulation of Survivable Armor Design Studies for IED Threats			**5. FUNDING NUMBERS**
6. AUTHOR(S) Douglas L. Kaminski			
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Naval Postgraduate School Monterey, CA 93943-5000			**8. PERFORMING ORGANIZATION REPORT NUMBER**
9. SPONSORING /MONITORING AGENCY NAME(S) AND ADDRESS(ES) N/A			**10. SPONSORING/MONITORING AGENCY REPORT NUMBER**
11. SUPPLEMENTARY NOTES The views expressed in this thesis are those of the author and do not reflect the official policy or position of the Department of Defense or the U.S. Government.			
12a. DISTRIBUTION / AVAILABILITY STATEMENT Approved for public release; distribution is unlimited			**12b. DISTRIBUTION CODE**

13. ABSTRACT (maximum 200 words)

Improvised Explosive Device (IED) is used as a strategic weapon of choice and continues to be a threat both globally and domestically. One of the deadly devices in this arsenal is the Explosively Formed Projectile (EPF). This study develops methodology for modeling and simulation of armor plates to survive EFP threats. The EFP effects are modeled as a pressure or blast wave using compressible isentropic conservation equations to get pressure loadings. The thermal effects are modeled as temperature intensities and resulting transient heat transfer analysis is conducted to obtain temperature distribution. The kinetic loads are modeled as high initial velocities applied to the plate. The combined mechanical and thermal loading is analyzed. The design space is generated for varying materials properties and thicknesses as parameters. Laminated composite and orthotropic composites are also used in addition to special high strength and high stiffness generic alloys. The analysis is done using both two-dimensional plate theories as well as three-dimensional transient dynamic analysis. The results are presented showing maximum stresses and deformations for different combinations of materials and thicknesses. The results also indicate the need to use three-dimensional analysis for designing survivable armor. Some recommendations are made for further studies.

14. SUBJECT TERMS Improvised Explosive Device Armor, Explosively Formed Projectile, Model and Simulation			**15. NUMBER OF PAGES** 85
			16. PRICE CODE
17. SECURITY CLASSIFICATION OF REPORT Unclassified	**18. SECURITY CLASSIFICATION OF THIS PAGE** Unclassified	**19. SECURITY CLASSIFICATION OF ABSTRACT** Unclassified	**20. LIMITATION OF ABSTRACT** UU

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. 239-18

THIS PAGE INTENTIONALLY LEFT BLANK

Approved for public release; distribution is unlimited

MODELING AND SIMULATION OF SURVIVABLE ARMOR DESIGN STUDIES FOR IED THREATS

Douglas L. Kaminski
Lieutenant, United States Navy
B.S., Naval Postgraduate School, 2008

Submitted in partial fulfillment of the
requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

from the

NAVAL POSTGRADUATE SCHOOL
March 2008

Author: Douglas L. Kaminski

Approved by: Ramesh Kolar
Thesis Advisor

Garth Hobson
Second Reader

Anthony J. Healy
Chairman, Department of Mechanical and Astronautical Engineering

THIS PAGE INTENTIONALLY LEFT BLANK

ABSTRACT

Improvised Explosive Device (IED) is used as a strategic weapon of choice and continues to be a threat both globally and domestically. One of the deadly devices in this arsenal is the Explosively Formed Projectile (EPF). This study develops methodology for modeling and simulation of armor plates to survive EFP threats. The EFP effects are modeled as a pressure or blast wave using compressible isentropic conservation equations to get pressure loadings. The thermal effects are modeled as temperature intensities and resulting transient heat transfer analysis is conducted to obtain temperature distribution. The kinetic loads are modeled as high initial velocities applied to the plate. The combined mechanical and thermal loading is analyzed. The design space is generated for varying materials properties and thicknesses as parameters. Laminated composite and orthotropic composites are also used in addition to special high strength and high stiffness generic alloys. The analysis is done using both two-dimensional plate theories as well as three-dimensional transient dynamic analysis. The results are presented showing maximum stresses and deformations for different combinations of materials and thicknesses. The results also indicate the need to use three-dimensional analysis for designing survivable armor. Some recommendations are made for further studies.

THIS PAGE INTENTIONALLY LEFT BLANK

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

THIS PAGE INTENTIONALLY LEFT BLANK

ACKNOWLEDGMENTS

I would like to thank Professor Ramesh Kolar for all the help he gave me. His extensive knowledge made it possible to quickly troubleshoot any model problems, making it possible to complete the present work on time.

THIS PAGE INTENTIONALLY LEFT BLANK

I. INTRODUCTION

A. IMPROVISED EXPLOSIVE DEVICE BACKGROUND

1. Lessons from Iraq

Improvised Explosive Devices (IEDs) are the single greatest cause of causalities and destruction of equipment in Iraq, Afghanistan, and elsewhere as shown in Figure 1. IEDs can be detonated in a multitude of ways from command detonation with a cell phone to a command detonation from a sensor to a simple trip wire. The resulting explosion produces a pressure wave that causes sever structural damage to a vehicle and can in some cases lift the vehicle off the ground and flip it over. More advanced IEDs also use an explosively formed projectile (EFP) to penetrate vehicle armor. The explosively formed projectile impacts the target with a molten mass of metal at velocities of 1.5 to 2 km per a second.

Figure 1. HUMV destroyed by IED

One solution to defend against IEDs and EFPs is military vehicles must be designed to mitigate the effects of EFPs to improve survivability of the vehicle by the use of advanced forms of armor. The old fashioned method of just adding inches of steel are not practical and would result in viability and maintenance issues with the vehicles. Additionally such a heavy vehicle would have a difficult time patrolling an urban area with its narrow streets and restrictive weight limits on the roads and bridges.

B. SURVIVABILITY

Survivability is simply the ability to of a military vehicle to go out on a mission and come back to do it again. Survivability is composed primarily of two elements, susceptibility and vulnerability. Susceptibility is the inability of a vehicle to avoid being damaged by the hostile environment. Vulnerability is the inability of a vehicle to withstand damage caused by the hostile environment. This paper will focus on mitigating the effects of an EFP by reducing the vulnerability of military vehicles by modeling a typical IED explosion and developing methodology for modeling and simulation of armor that will withstand that attack allowing the crew to evacuate and fight on. A simple IED attack looks like Figure 2.

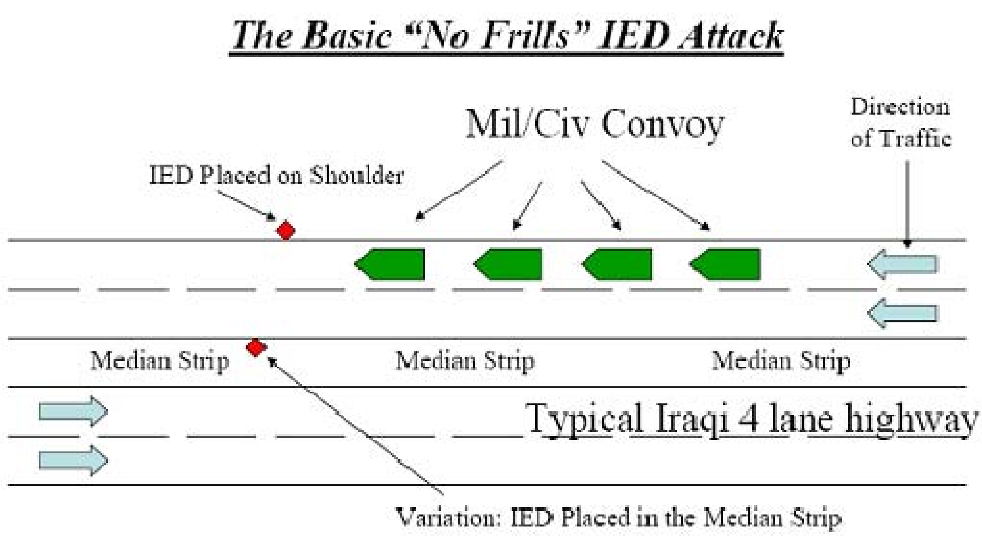

Figure 2. Diagram of simple IED attack.

1. Explosively Formed Projectile

An Explosively Formed Penetrator (EFP) is a class of IEDs with a shaped warhead designed to penetrate armor effectively at stand-off distances. An EFP system comprises three major components: 1) an arming device, 2) a trigger device and 3) the explosive device. Figure 3 shows a diagram for an EFP. The explosive device component consists of four elements: a concave metal liner (warhead), case, base plate, and explosive charge (propellant). The case holding the explosive is generally cylindrical, fabricated from commonly available materials (e.g. PVC pipe, steel pipe), with the forward end closed by a concave copper or steel disk-shaped liner to create a penetrator. Generally military grade explosive is loaded behind the metal liner to fill the casing. A blasting cap initiator is placed through a hole centered in the base plate. Upon detonation, the explosive projects the liner to form a projectile in a direction in line with the EFP at a speed well over one kilometer per a second, depending on the design and the type of explosive used. Detonation is controlled by command wire, radio control (RC), or remote arming with trigger. EFPs can be deployed singly, in pairs, or in arrays, and camouflaged in painted foam to look similar to natural rocks or with man made artifacts (e.g. trash) depending on the tactical situation.

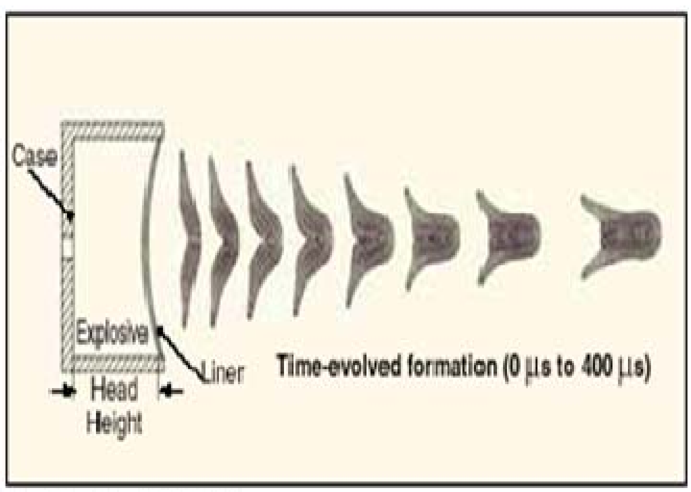

Figure 1. Formation of an EFP warhead

Figure 3. Diagram of EFP detonation and projectile shaping.

In order for the crew of a vehicle to survive and evacuate, the vehicle's armor would have to with stand the resulting pressure wave and prevent the EFP from entering the crew cabin.

II. THEORY

A. THEORETICAL MODEL OF IED EXPLOSION AND IMPACT

1. Pressure Wave

In this section a simple IED explosion will be examined from the point of explosion to the point of impact, on a rectangular plate of armor. When an IED is detonated; the resulting explosion produces a pressure wave. Typically the explosion is modeled [1] as a compressible, time-dependent, conservation equations for the mass density, momentum density and energy density. These equations, written in their Eulerian form and in one-dimensional spherical coordinates are [1]:

$$\frac{\partial \rho}{\partial t} = -\frac{1}{r^2}\frac{\partial r^2 \rho v}{\partial r} \tag{1}$$

$$\frac{\partial \rho v}{\partial t} = -\frac{1}{r^2}\frac{\partial r^2 \rho v v}{\partial r} - \frac{\partial p}{\partial r} \tag{2}$$

$$\frac{\partial e}{\partial t} = -\frac{1}{r^2}\frac{\partial r^2 e v}{\partial r} - \frac{1}{r^2}\frac{\partial r^2 v p}{\partial r} \tag{3}$$

where ρ is the density, v the fluid velocity, p the pressure and e the volumetric density of the total energy, v and t are the radial coordinate and time respectively. The total energy density e is defined as the sum of the internal energy and kinetic energy. Using equation 4 the work done by the gas during an isentropic expansion can be expressed as

$$W_{is} = p_1 V_1 \frac{1}{\gamma_1 - 1}\left[1 - \left(\frac{p_0}{p_1}\right)^{(\gamma_1 - 1)/\gamma_1}\right] \tag{4}$$

where P_0 is the ambient temperature and P_1 is the expected over pressure.

The peak pressure p_s and distance R are non-dimensionalized using Sachs's scaling relationships.[1]

$$\overline{p_s} = \left(p_s - p_o\right) / p_o \tag{5}$$

$$\overline{R} = \frac{R}{\left(\frac{W_{is}}{p_o}\right)^{1/3}} \tag{6}$$

The peak over pressure can then be calculated using equation 6.

$$p_s - p_o = \overline{p_s} p_o \tag{7}$$

This resulting overpressure is applied as a pressure force on all subsequent simulations of the IED effects on the armor plate..

2. Explosively Formed Projectile

In this study, the EFP effects are modeled in two steps. The thermal effects are modeled as high temperature flux on the plate. The kinetic energy is modeled as an applied initial velocity to the plate. The equations of motion may be expresses as follows

$$[M]\left\{\ddot{x}\right\} + [K]\left\{x\right\} = \{P\} \tag{8}$$

where [M] is the mass matrix, [K] the stiffness matrix, {x} the displacement vector, and {P} the loading vector. Direct numerical integration method is used in computing the response to the combined loads. [8]

B. MATERIAL ENERGY ABSORPTION

1. Deformation

Materials absorb mechanical energy through the process of deformation. At first the deformation is elastic and any changes in shape will revert to the original shape once the load is removed. After elastic deformation comes inelastic; any changes in shape will be permanent. The amount of deformation, both elastic and inelastic, that a material can undergo is called ductility. The more ductile the material the more it can deform and the more energy it will absorb.

The work done by external forces in causing deformation is stored within the body in the form of strain energy. In an ideal elastic process, no dissipation of energy takes place and all the stored energy is recoverable upon unloading.

The temperature distribution is used to conduct a transient heat transfer analysis resulting in temperature distributions in the plate as a function of time. In the next step, these temperatures are applied as thermal loads together with blast loads and initial velocity to perform a transient dynamic analysis resulting in displacements, strains, and stresses caused in the armor plate.

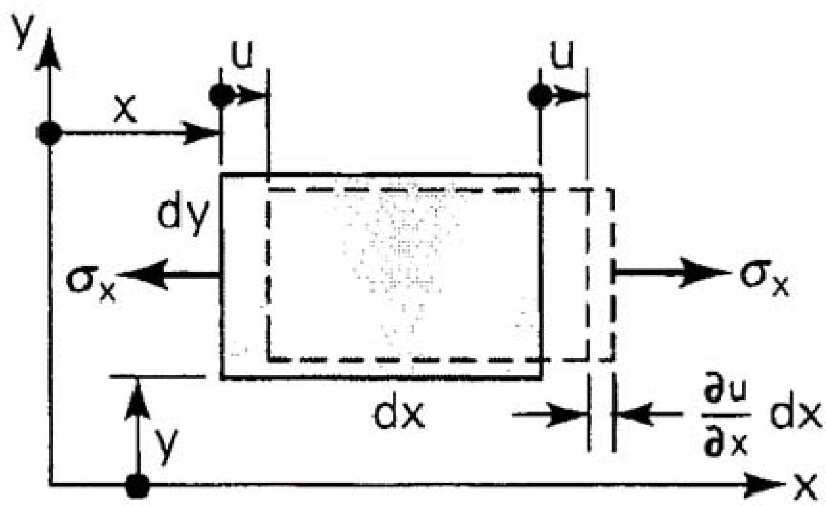

Figure 4. Displacement under uniaxial stress from [2].

The net work done on an element can be calculated by [2]:

$$dW = dU = \int_0^{\varepsilon_x} \sigma_x d\left(\frac{\partial u}{\partial x}dx\right)dydz = \int_0^{\varepsilon_x} \sigma_x d\varepsilon_x \left(dxdydz\right) = \tag{9}$$

Note that dW is the work done on dx dy dz, and dU is the corresponding increase in strain energy. Designating the strain energy per a unit volume as U_0, for a linearly elastic material in one dimension, we have

$$U_0 = \int_0^{\varepsilon_x} \sigma_x d\varepsilon_x = \int_0^{\varepsilon_x} E\varepsilon_x d\varepsilon_x \tag{10}$$

After integration, Equation 10 yields:

$$U_0 = \frac{1}{2} E \varepsilon_x^2 = \frac{1}{2} \sigma_x \varepsilon_x^2 = \frac{1}{2E} \sigma_x^2 \tag{11}$$

This quantity represents the shaded area in Figure 5. The units of U_0, strain energy density, in SI are joules per cubic meter (J/m^3), or pascals (Pa).

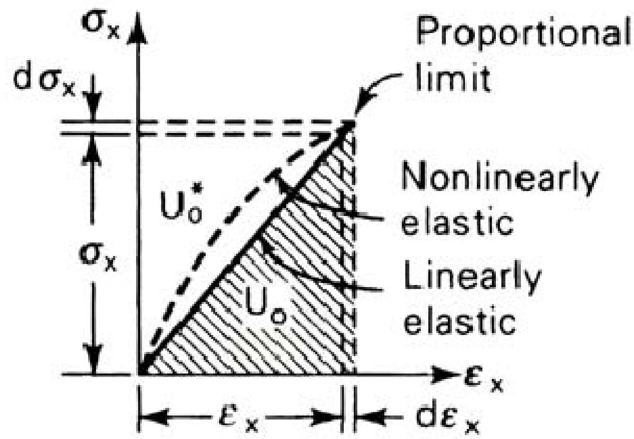

Figure 5. Work done by uniaxial stress from [2].

The shaded area under the stress-strain curve represents the measure of a material's ability to absorb energy without permanent deformation, and is defined as the modulus of resilience. The area under the entire stress-strain curve provides a measure of a material's ability to absorb energy up to the point of fracture, and is defined as the modulus of toughness.

Shear deformation and the elastic strain energy associated with it is now analyzed by considering an element of thickness dz subject to only shearing stress τ_{xy}, see Figure 6. Note that the shear force τ_{xy} dxdz causes a displacement of γ_{xy} dy. The strain energy due to shear is $\frac{1}{2}(\tau_{xy} dxdz)(\gamma_{xy} dy)$, where the factor of ½ arises because the stress varies linearly with the strain.

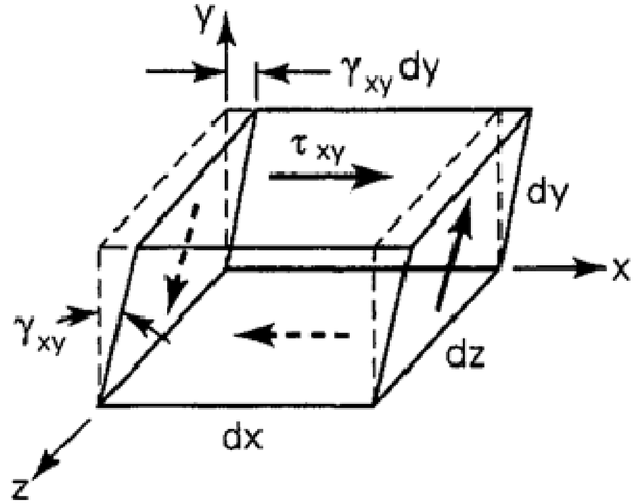

Figure 6. Deformation due to pure shear from [2].

The strain energy is.

$$U_0 = \frac{1}{2}\tau_{xy}\gamma_{xy} = \frac{1}{2G}\tau_{xy}^2 = \frac{1}{2}G\gamma_{xy}^2 \qquad (12)$$

Because the work done by τ_{xy} accompanying perpendicular strains γ_{yz} and γ_{xz} is zero, the total strain energy density done by shear alone is found by superposition of three terms identical in the form of Equation 11.

$$U_0 = \frac{1}{2}\left(\tau_{xy}\gamma_{xy} + \tau_{yz}\gamma_{yz} + \tau_{xz}\gamma_{xz}\right) \qquad (13)$$

C. TRANSIENT VIBRATION

The force of the explosion from the IED will impact on the armor plate for very short finite time with a force of large magnitude. This impact of force application is best modeled as an Impulse Excitation [3].

$$\hat{F} = \int F(t)dt \qquad (14)$$

Because F dt = m dv, the impulse acting on the armor will result in sudden change in the armor's velocity equal to $\hat{F}/m$ without appreciable change in its displacement.

Under free vibration, the undamped single degree of freedom spring-mass system with initial displacement of $x(0)$ and initial velocity of $\dot{x}(0)$ given by the equation

$$x(t) = \frac{\dot{x}(0)}{\omega_n}\sin\omega_n t + x(0)\cos\omega_n t \tag{15}$$

Hence, the response of a spring mass system initially at rest and excited by an impulse $\hat{F}$ is

$$x = \frac{\hat{F}}{m\omega_n}\sin\omega_n t = \hat{F}h(t) \tag{16}$$

where

$$h(t) = \frac{1}{m\omega_n}\sin\omega_n t \tag{17}$$

is the response to a unit impulse.

D. THERMAL ANALYSIS

1. Heat Transfer

The impact of the copper projectile from an EFP will result in thermal loads to the armor plate. For the purpose of this research the thermal effects of an EFP will be modeled as an initial temperature of 1400K acting at the same point as the initial velocity. The armor design requirements need to dissipate this heat energy effectively and efficiently. We assume that heat will transfer throughout the armor by conduction in accordance with Fourier's law [4]

$$q'' = -k\nabla T = -k\left(i\frac{\partial T}{\partial x} + j\frac{\partial T}{\partial y} + k\frac{\partial T}{\partial z}\right) \tag{18}$$

where q'' is the heat flux, k is the thermal conductivity, (k is negative to indicate the flow of heat from hot to cold) T is the temperature, and ∇ is the gradient operator

2. Thermal Stresses

The resulting increase in temperature throughout the armor as a result of the temperature flux will cause the armor to expand, via thermal expansion

$$\varepsilon_t = \alpha \Delta T \tag{19}$$

where ε_t is the thermal strain, α is the coefficient of linear thermal expansion, and ΔT is the change in temperature. The total strains on the armor plate in two dimensions would then become [2]

$$\begin{aligned}
\varepsilon_x &= \frac{1}{E}\left(\sigma_x - \upsilon\sigma_y\right) + \alpha\Delta T \\
\varepsilon_y &= \frac{1}{E}\left(\sigma_y - \upsilon\sigma_x\right) + \alpha\Delta T \\
\gamma_{xy} &= \frac{\tau_{xy}}{G}
\end{aligned} \tag{20}$$

Notice that the thermal expansion doesn't result in shear distortion for isotropic materials, where as in a composite material this will not hold true.

Since the armor is constrained on all its edges the resulting stress in terms of strain components become [2]

$$\begin{aligned}
\sigma_x &= \frac{E}{1-\upsilon^2}\left(\varepsilon_x + \upsilon\varepsilon_y\right) - \frac{E\alpha\Delta T}{1-\upsilon} \\
\sigma_y &= \frac{E}{1-\upsilon^2}\left(\varepsilon_y + \upsilon\varepsilon_x\right) - \frac{E\alpha\Delta T}{1-\upsilon} \\
\tau_{xy} &= G\gamma_{xy}
\end{aligned} \tag{21}$$

In equation (21), the first term corresponds to the mechanical loads (blast, initial velocity) and the second term corresponds to the thermal effects.

THIS PAGE INTENTIONALLY LEFT BLANK

III. FINITE ELEMENT FORMULATION

Finite elements represent spatial descretization of a continuum. As such, however, they do not immediately impose nonlinearity. When nonlinearity has to be taken into account for larger displacements and /or stress, a numerical model poses new dimensions to the descretization in addition to the spatial descretization. That is, the descretization is applied to time, load, and material properties by using piecewise linear curves. While descretization allows approximate solutions by numerical methods, it introduces numerous mathematical singularities which may complicate computational processes. Fortunately, the efficiency of modern digital computers makes it feasible to apply complicated computational procedures to the complex systems of engineering problems.

The variational principle renders the system governing equilibrium equations when applied to a functional $\prod$, representing a total potential of a continuum

$$\prod = U - W \tag{22}$$

where U is the strain energy of the system and W is the potential energy of the external loads. The equilibrium equations can be obtained by invoking the principle of virtual work or the Ritz method

$$\delta \prod = 0 \text{ or } \frac{\partial \prod}{\partial \{u\}} = 0 \tag{23}$$

which implies that the total potential of the system must be stationary with respect to the state variables (displacement) for equilibrium to be ensured. The functional $\prod$ is so called because it involves the integral of implicit functions of the state variables, $\{\text{u}\}$.

Considering a three-dimensional continuum for a nonlinear problem, the stationarity condition results in

$$\int_V \sigma_{ij} \delta \dot{\varepsilon}_{ij} \, dV = \int_V b_i \delta \dot{\varepsilon}_i \, dV + \int_S t_i \delta \dot{\varepsilon}_i \, dS + \sum_i p_i \delta \dot{u}_i \tag{24}$$

Where the dots and δ denote infinitesimal increments and arbitrary variations, respectively.

In this section a brief overview of the finite element approach for modeling transient thermo-mechanical response is presented [8].

The finite element method can be characterized by the following features distinguished from the conventional Ritz methods or the matrix method for frame structures:

- The whole region of the system is divided into numerous subdomains, called finite elements, which have simple geometrical shapes.
- The variational process is limited to each finite element, which aggregates into a whole region when assembled.
- The admissible displacement field within each element, $\left\{\tilde{u}\right\}$, can be expressed in terms of nodal displacements using interpolation functions known as shape functions, N,

$$\left\{\tilde{u}\right\} = \left[N\right]\left\{u\right\} \tag{25}$$

where $\left\{u\right\}$ is a displacement vector consisting of all nodal points of the element.

The strain displacement relations for the element can then be established in terms of nodal displacements using the shape functions in Equation 26

$$\left\{\dot{\varepsilon}\right\} = \left[B\right]\left\{\dot{u}\right\} \tag{26}$$

where

$$\left\{\dot{\varepsilon}\right\} = \left\langle \dot{\varepsilon}_x \, \dot{\varepsilon}_y \, \dot{\varepsilon}_z \, \dot{\gamma}_{xy} \, \dot{\gamma}_{yz} \, \dot{\gamma}_{zx} \right\rangle \tag{27}$$

and the element matrix [B] consists of derivatives of the shape functions, evaluated at the current deformed geometry. MSC/NASTRAN employs an approximated Lagrangian approach for geometric nonlinear problems, by which linear strains are computed in the

updated element coordinate system in order to eliminate the effects of the rigid body rotation but the equilibrium is established at the final position in the stationary coordinate system.

Equilibrium equations for an element may be obtained by reducing Equation 24 after the substitution of Equations 25 and 26, based on the small deformation theory. Then the element boundary stresses are equivalent to the nodal forces which balance the applied external loads

$$\{F\}^e = \{P\}^e \tag{28}$$

with

$$\{F\}^e = \int_V [B]^T \{\sigma\}\, dV \tag{29}$$

and

$$\{P\}^e = \int_V [N]^T \{b\}\, dV + \int_s [N_s]^T \{t\}\, dS + \{p\} \tag{30}$$

where $[N_s]$ is an appropriate interpolation function for the traction force. Notice that the equilibrium equation for an incremental load may be expressed as

$$\left\{\dot{F}\right\} = \int_V [B]^T \left\{\dot{\sigma}\right\} dV = \left\{\dot{P}\right\} \tag{31}$$

where $\left\{\dot{\sigma}\right\}$ should be components of co-rotational stress which is independent of a rigid body rotation.

The element stiffness matrix can be obtained by substituting the constitutive relations into Equation 32,

$$\left\{\dot{\sigma}\right\} = [D]\left\{\dot{\varepsilon}\right\} \tag{32}$$

where

$$\{\sigma\}^T = \left\langle \sigma_x \sigma_y \sigma_z \tau_{xy} \tau_{yz} \tau_{zx} \right\rangle \tag{33}$$

and $[D]$ is a tangent material matrix. The nodal forces of an element can then be expressed as

$$\left\{\dot{F}\right\}^e = \int_V [B]^T \left\{\dot{\sigma}\right\} dV = [K]^e \left\{\dot{u}\right\} \tag{34}$$

where the element stiffness is

$$[K]^e = \int_V [B]^T [D][B] dV \tag{35}$$

Notice that this expression represents an element stiffness due to material stiffness without geometric nonlinear effects.

The equilibrium must be satisfied in the whole region throughout the complete history of load application. Equilibrium equations for the global discrete system are obtained when all the elements are assembled,

$$\sum_M \int_V [B]^T \{\sigma\} dV = \sum_M \{P\}^e \tag{36}$$

where $\sum$ over M denotes a summation over all elements. For the incremental process, the equilibrium equations may be rewritten as

$$\sum_M \int_V [B]^T \left\{\sigma - \sigma^0\right\} dV = \{\vartriangle P\} \tag{37}$$

with

$$\{\vartriangle P\} = \sum_M \{P\}^e - \sum_M \int_V [B]^T \left\{\sigma^0\right\} dV \tag{38}$$

where $\left\{\sigma^0\right\}$ represents an initial stress of the stress state at the preceding load step.

Because of the approximations involved in the interpolation functions, the finite element model provides an approximate solution even if the equilibrium Equation 37 is satisfied exactly. Consequently, the differential equations of equilibrium are not satisfied exactly even for linear problems, but the error decreases as the finite element mesh is refined. This convergence condition is required and ensured by element formulations with regard to the element convergence criteria. The convergence, however, may not be monotonic due to non-conforming elements or reduced integration. By virtue of the non-

conforming elements and the reduced integration, the finite element model will have added flexibility and compensate the stiffening effects by the displacement method. With a displacement approach, the finite element model is generally known to produce a stiffer structure than in reality.

THIS PAGE INTENTIONALLY LEFT BLANK

IV. PROCEDURE

In the present study the engineering analysis tools by MSC software consisting of PATRAN and NASTRAN is used for modeling and simulation. MSC/PATRAN is utilized to model the geometry of the armor plate, the materials, thickness properties, temperature fluxes, pressure loads, initial conditions and boundary conditions. Here in procedures and parameters are also selected in this module.

The input deck for MSC/NASTRAN is generated and edited if necessary. The MSC/NASTRAN solver is used to obtain the solution from the simulation. The MSC/PATRAN is invoked again to post-process the results and analyze the behavior of the armor design.

In the remains of this chapter, detailed procedural steps to model and simulate the transient thermal and transient thermo-elastic analysis is presented.

The armor plate is modeled as a 3m x 2m plate with all edges clamped. The temperatures, initial velocities, pressure loading, material properties, thickness properties, composite laminate properties and lay-up are used as design parameters.

The armor plate was modeled as follows.

- 2-D plate. In this case the armor plate is modeled as a plate/shell structure using QUAD4 elements of the analyzing procedure.
- 2-D laminated plate. In this case the armor plate is modeled as a laminated composite plate with each laminate modeled as an orthotropic layer.
- 3-D plate. In this case the armor plate is modeled as a three dimensional structure using higher order tetrahedral elements (tet10) of the analysis procedure.

It may be noted that for laminated composite plate analysis only blast loads and kinetic impact loads were considered without the thermal loads due to software limitations. Please see the Appendix for detailed procedures.

V. RESULTS

In this chapter, the results for thermal analysis, thermo-elastic analysis under thermal, blast and kinetic loads are presented. It is noted that transient heat conduction analysis is performed to obtain temperature as a function of time. These temperatures are then applied to the finite elements as thermal loads. The mechanical loads are superimposed with thermal loads. The kinetic loads are incorporated as prescribed initial velocities. A transient analysis using direct solution is used to obtain the resulting dynamic response of the armor plate. Initial linear free vibration analysis is done to select appropriate time step for direct solution.

A. KINETIC LOADING RESULTS

1. Kinetic Loading Results of Various Materials at 10cm Thickness in 2D

The kinetic loads applied to the armor plate include the pressure wave evenly distributed across the top surface of the plate, and the initial velocity applied to the center of the plate. The results are presented in the following figures, where the the stresses are in Pascals (Pa) and deflections are in meters (m).

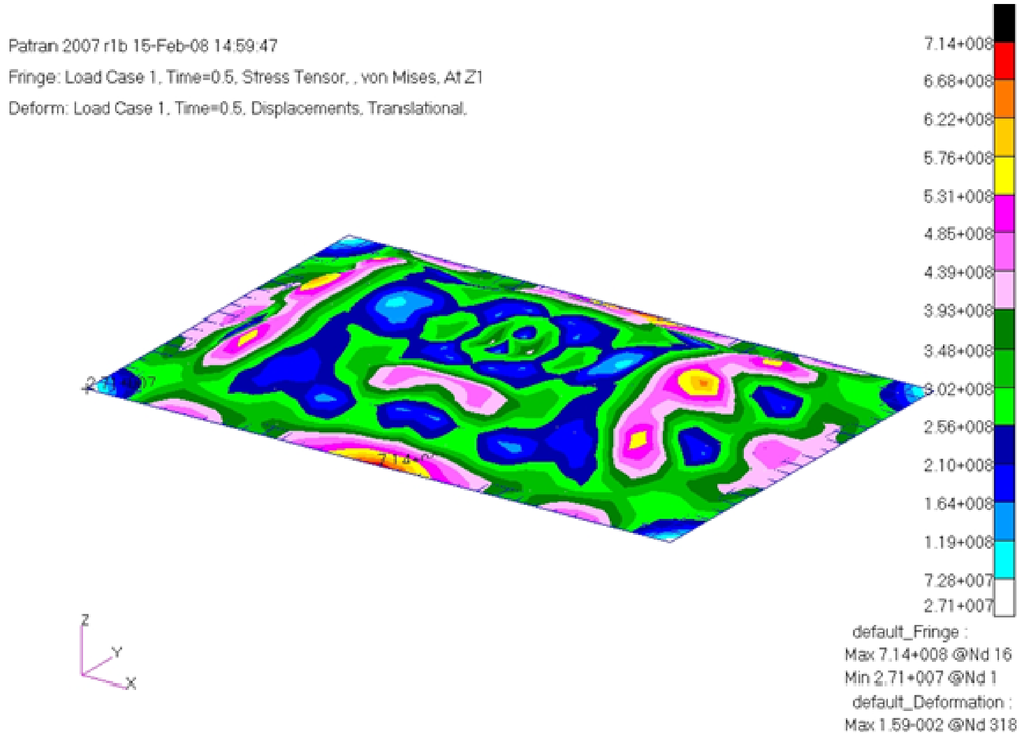

Figure 7. Stress distribution across an aluminum plate.

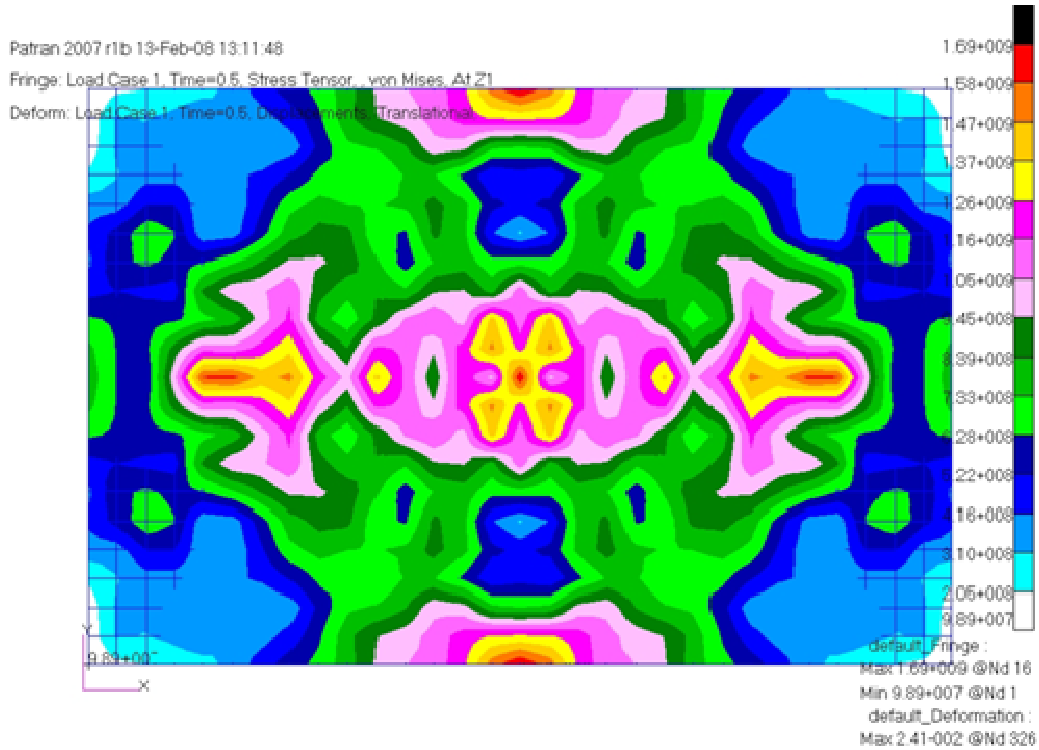

Figure 8. Stress distribution across a steel plate.

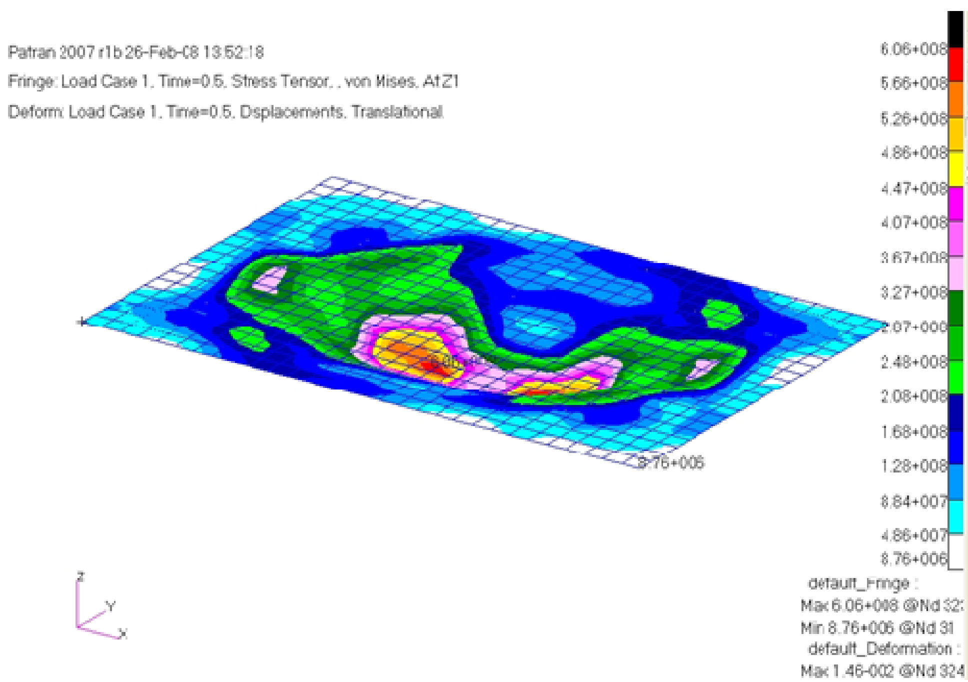

Figure 9. Stress distribution across a high-modulus graphite epoxy plate, homogeneous.

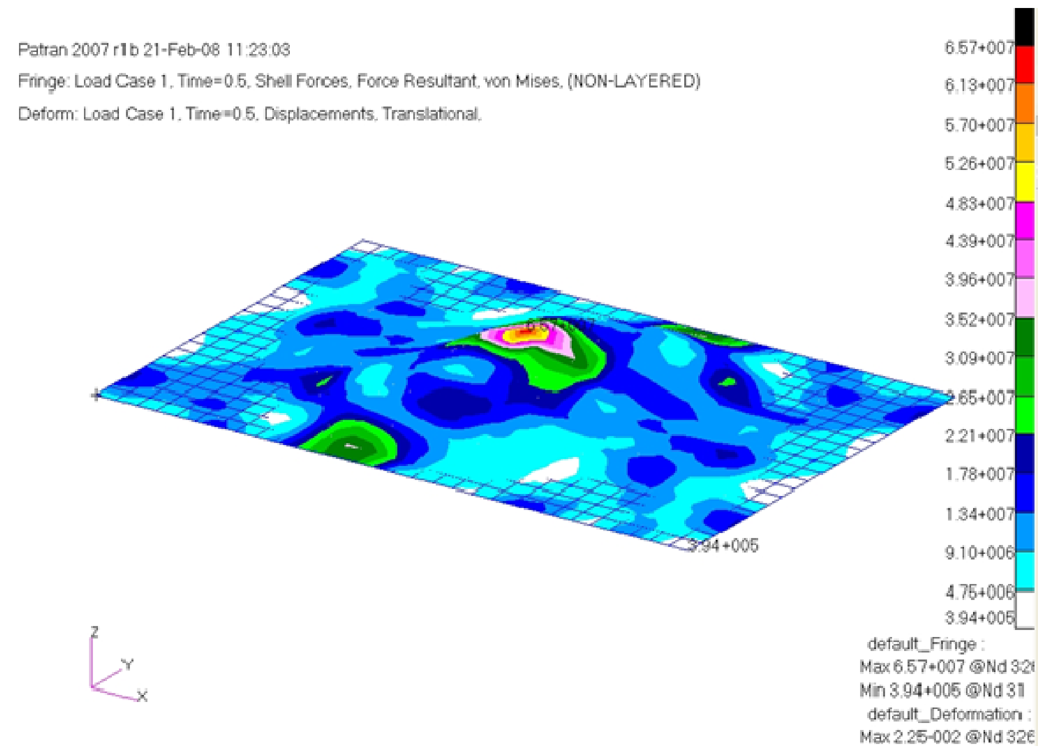

Figure 10. Stress distribution across a high-strength graphite epoxy plate, homogeneous.

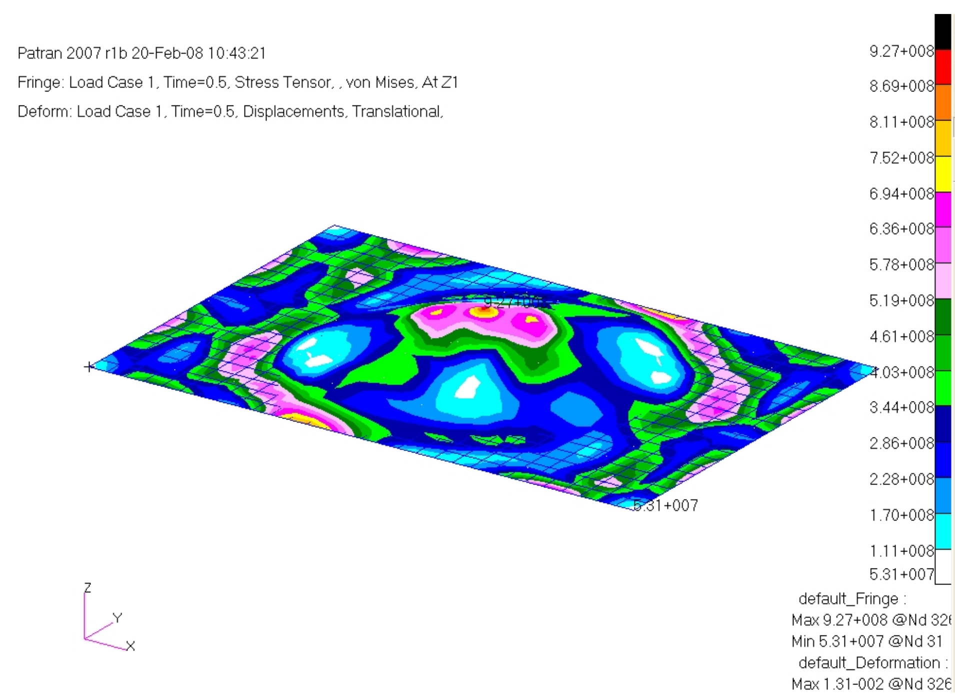

Figure 11. Stress distribution across a multi-material plate, homogeneous.

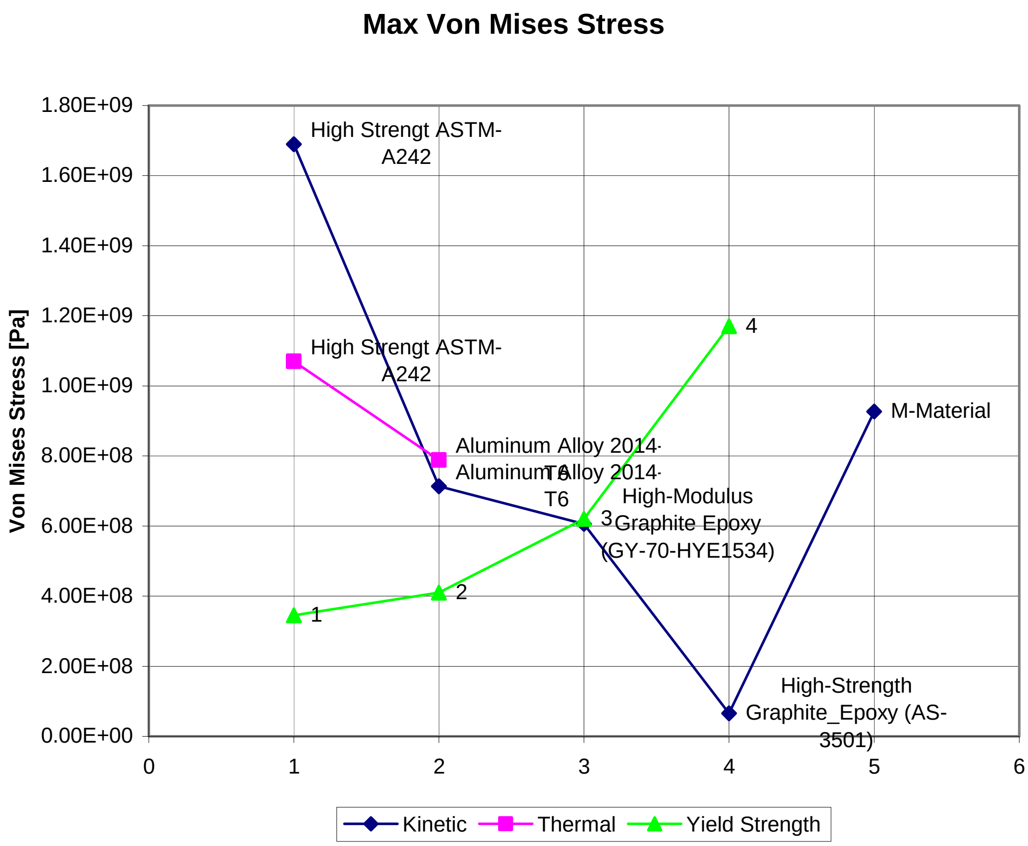

Figure 12. Plot of Von Mises Stresses of tested materials.

The results of various simulations are summarized in Figure 12. The x-axis is used to identify different materials and y-axis indicates Von-Mises stresses. It can be seen that the ASTM-A242 steel had the highest stress while the graphite composites had the lowest with the high-strength graphite being one order of magnitude lower than the rest.

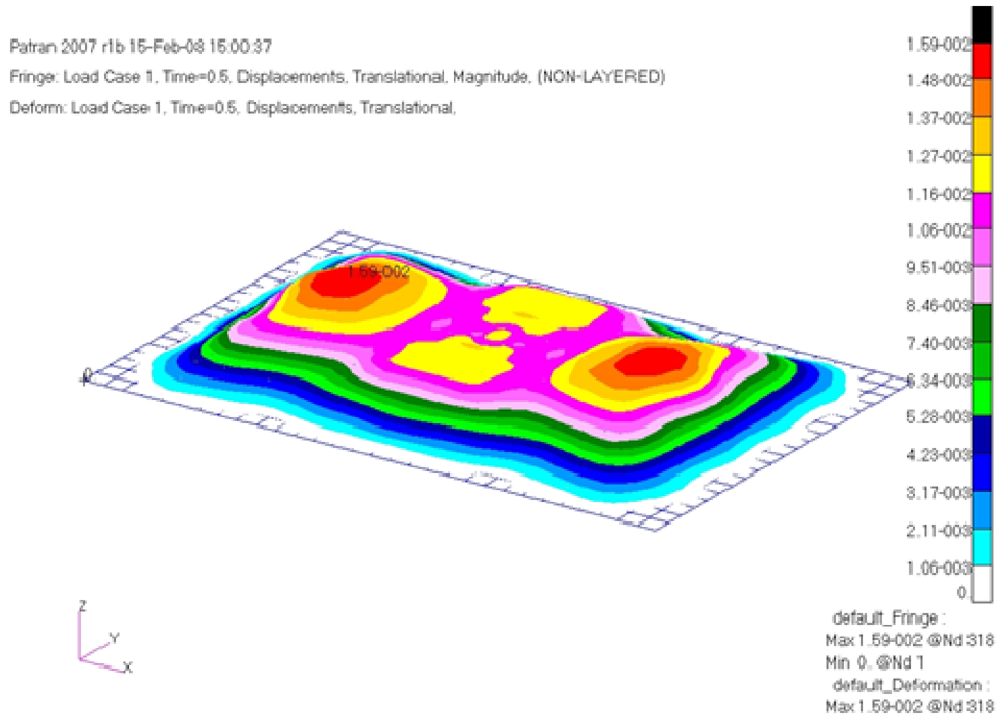

Figure 13. Deflections across an aluminum plate.

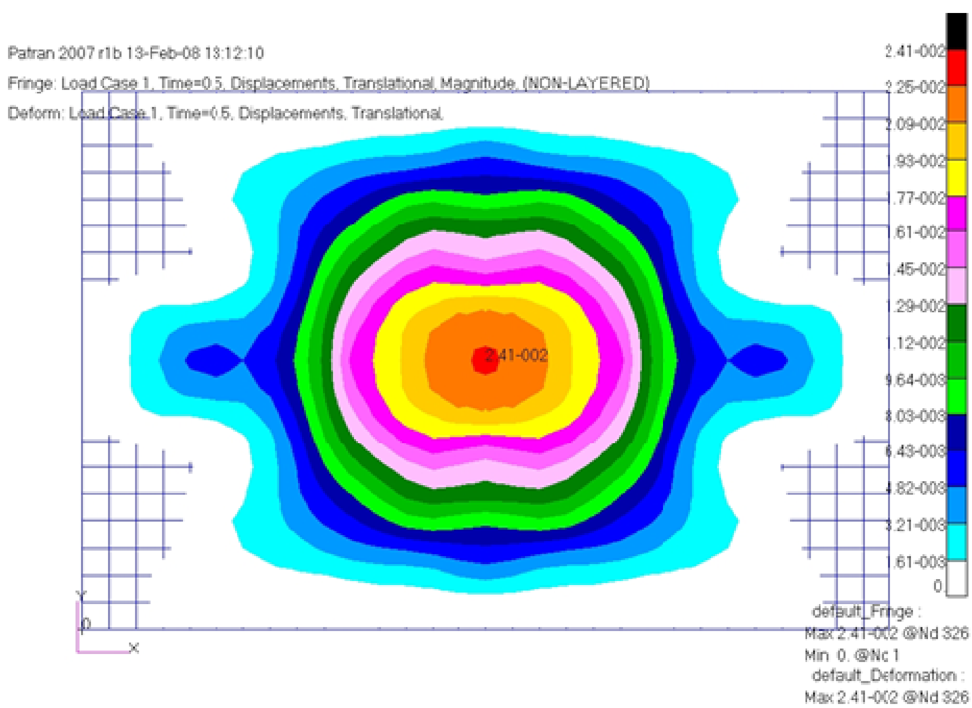

Figure 14. Deflections across a steel plate.

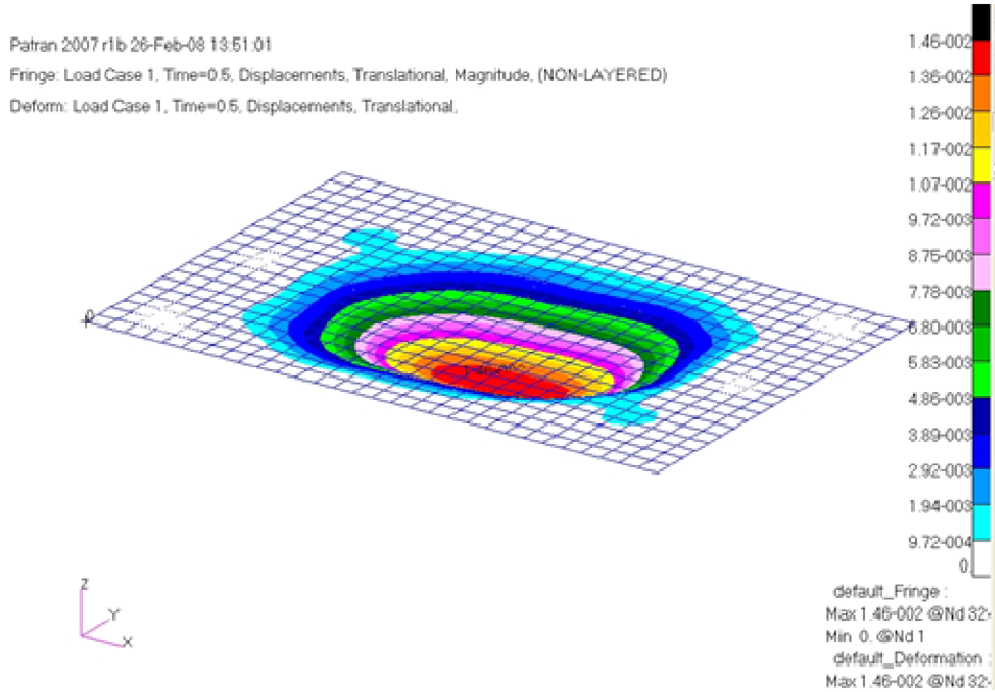

Figure 15. Deflections across a high-modulus graphite epoxy plate, homogeneous.

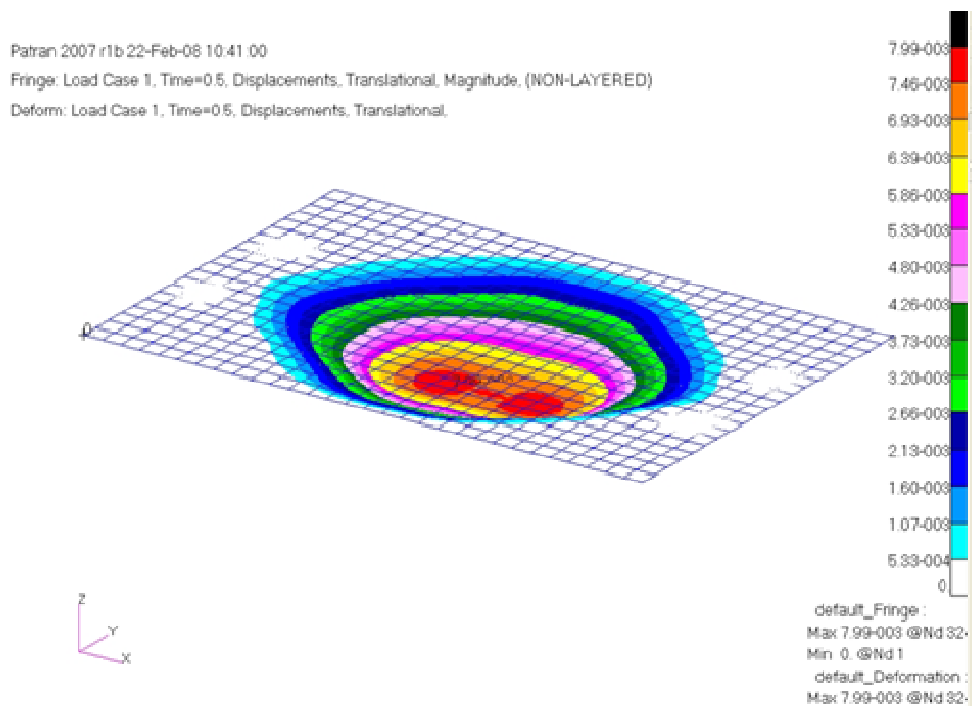

Figure 16. Deflections across a high-strength graphite epoxy plate, homogeneous.

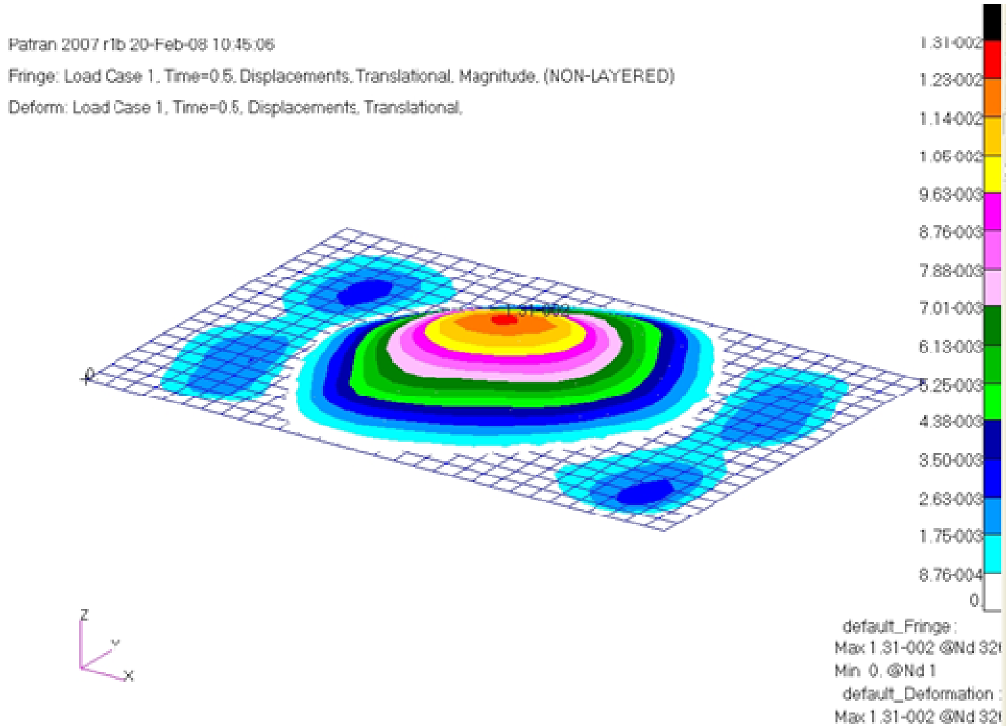

Figure 17. Deflections across a multi-material plate.

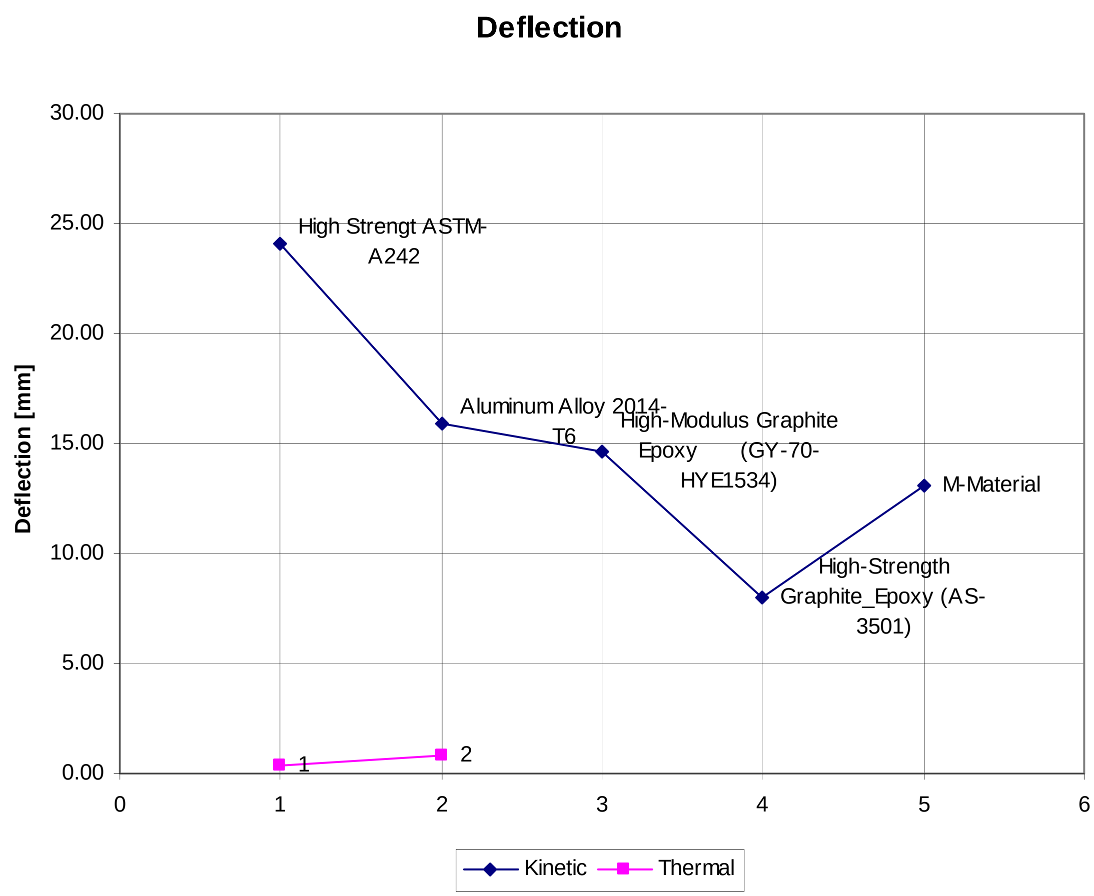

Figure 18. Plot of deflections of tested materials.

The deflections(max) for avious materials is summarized in Figure 18. It is seen the stresses using steel has the greatest deflections while the graphite composites have the least and the high-strength composite is an order of magnitude lower. The composites in this case were analyzed as orthotropic material in NASTRAN.

2. Kinetic Loading Results of Steel at Varying Thickness in 2D

The results for the armor plate under kinetic loading are presented here. The maximum Von Mises stress and maximum deflection of steel as varying thickness was found to help determine the optimum thickness for the armor plate. The kinetic loads didn’t change only the thickness was changed in the model.

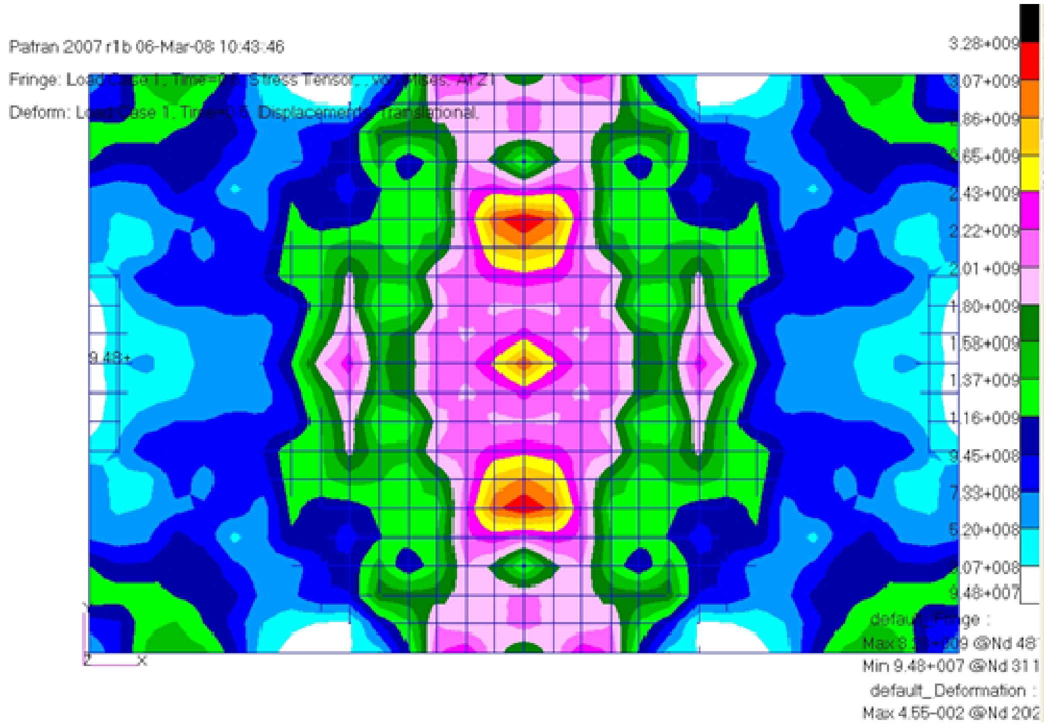

Figure 19. Stress distribution across a 4 cm thick steel plate.

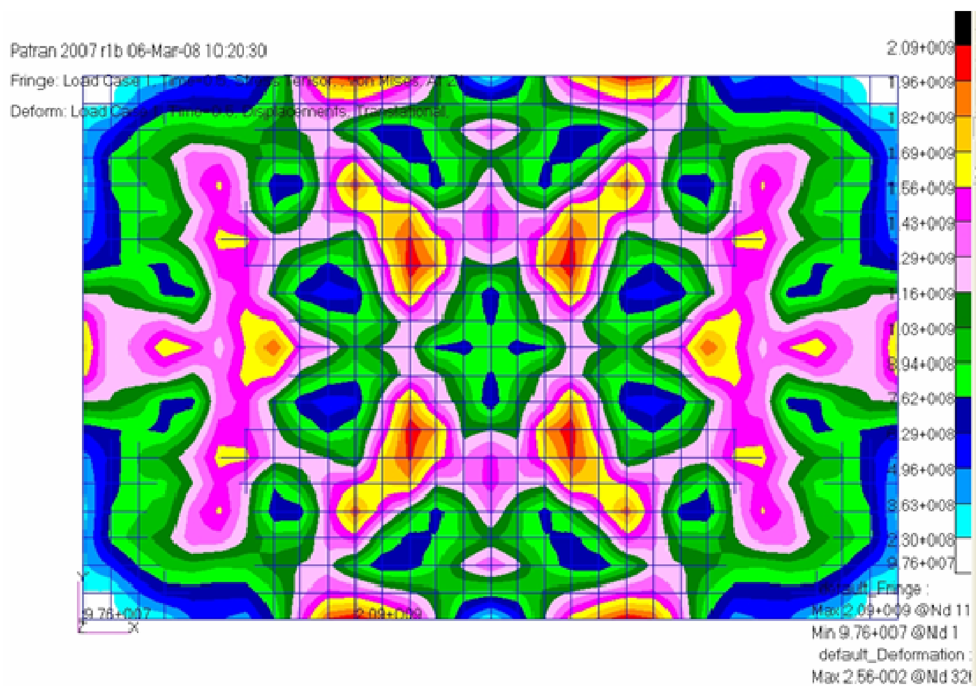

Figure 20. Stress distribution across a 6 cm thick steel plate.

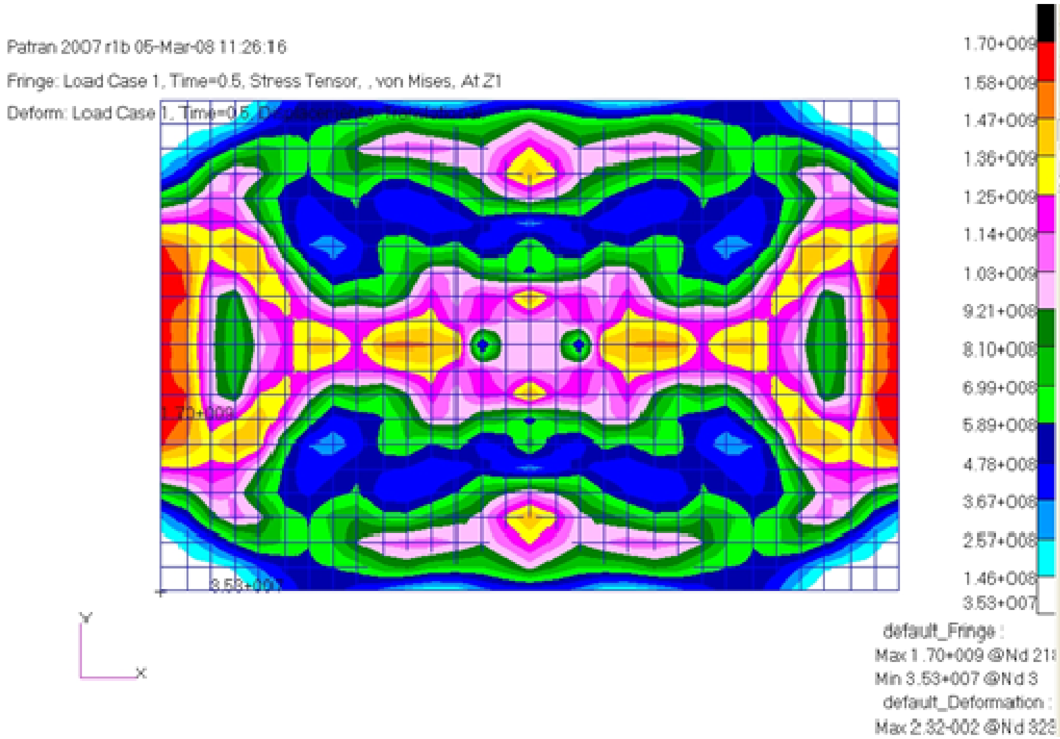

Figure 21. Stress distribution across an 8 cm thick steel plate.

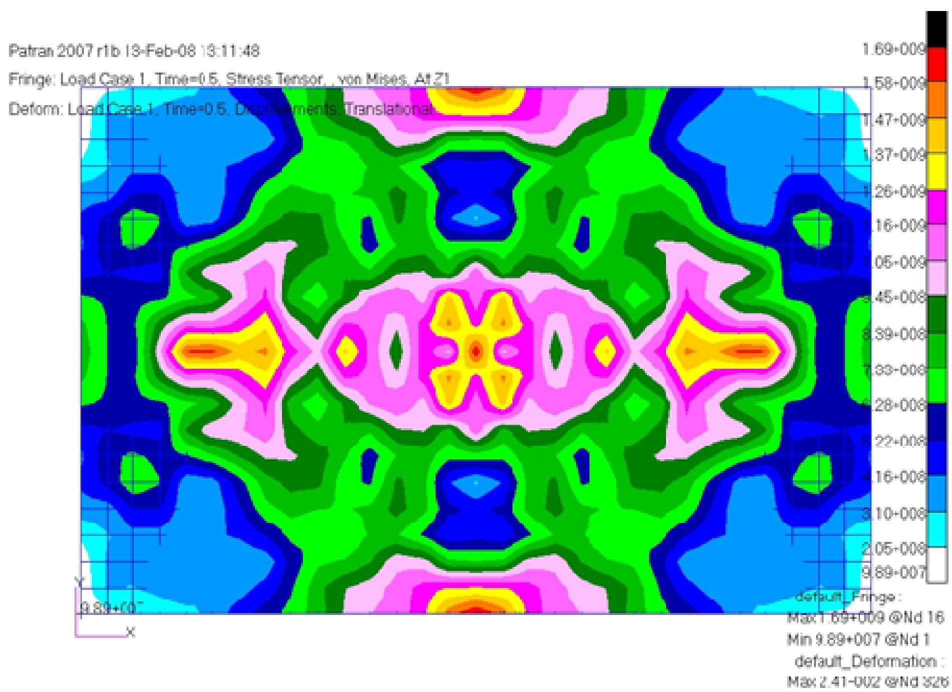

Figure 22. Stress distribution across a 10 cm thick steel plate.

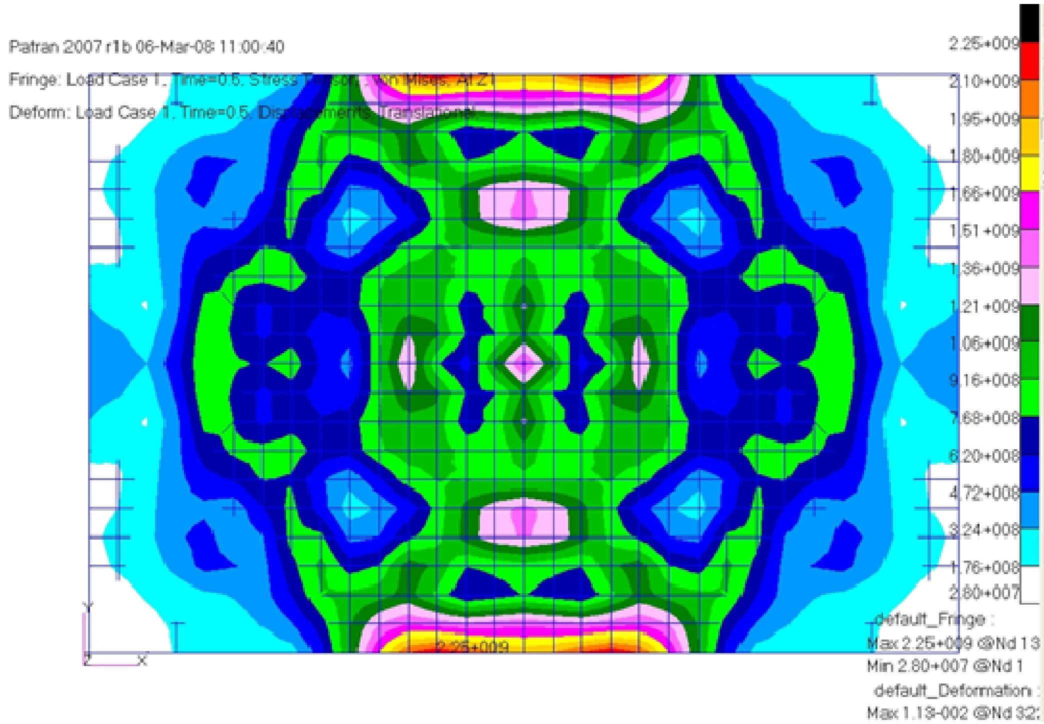

Figure 23. Stress distribution across a 12 cm thick steel plate.

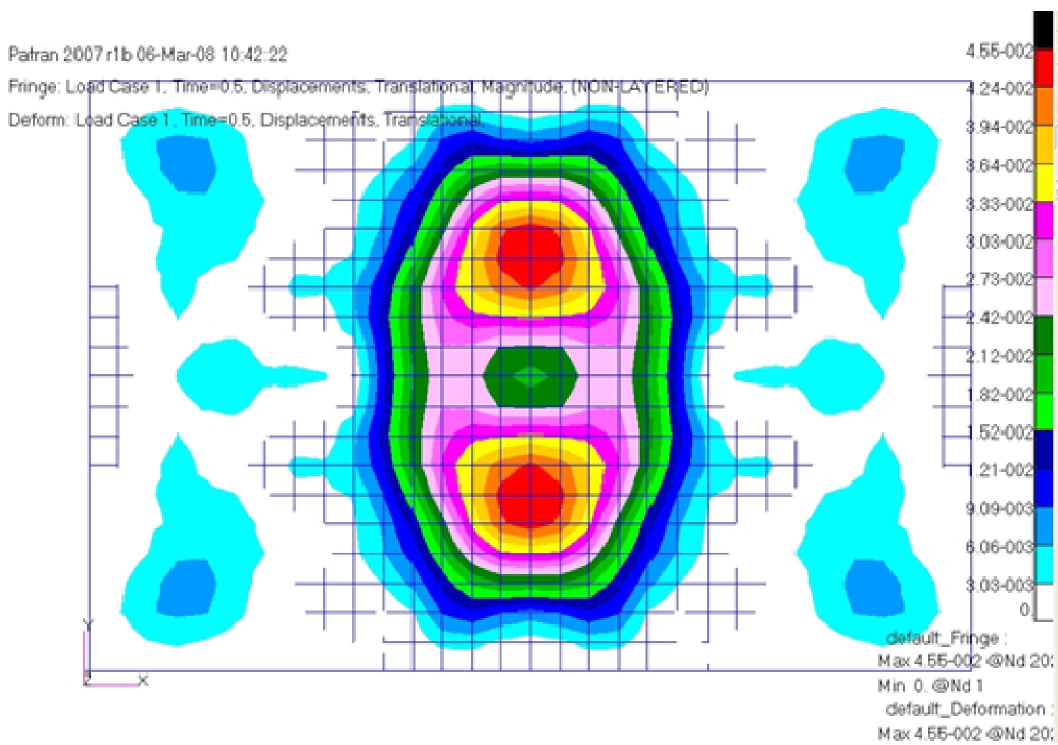

Figure 24. Displacements across a 4 cm thick steel plate.

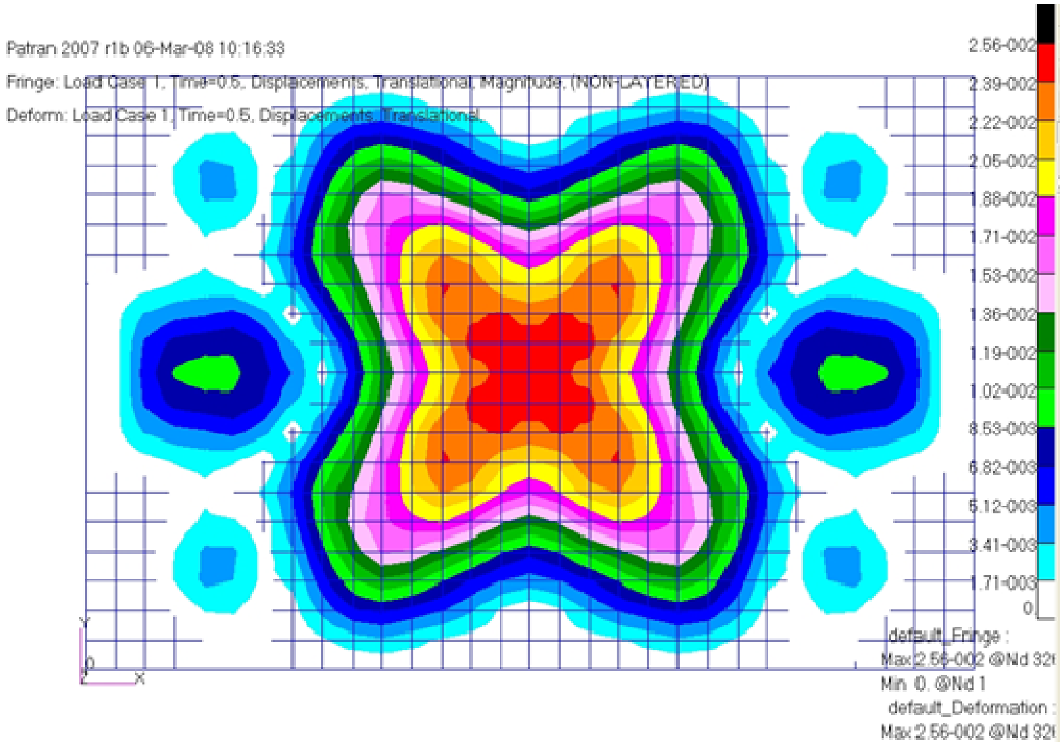

Figure 25. Displacements across a 6 cm thick steel plate.

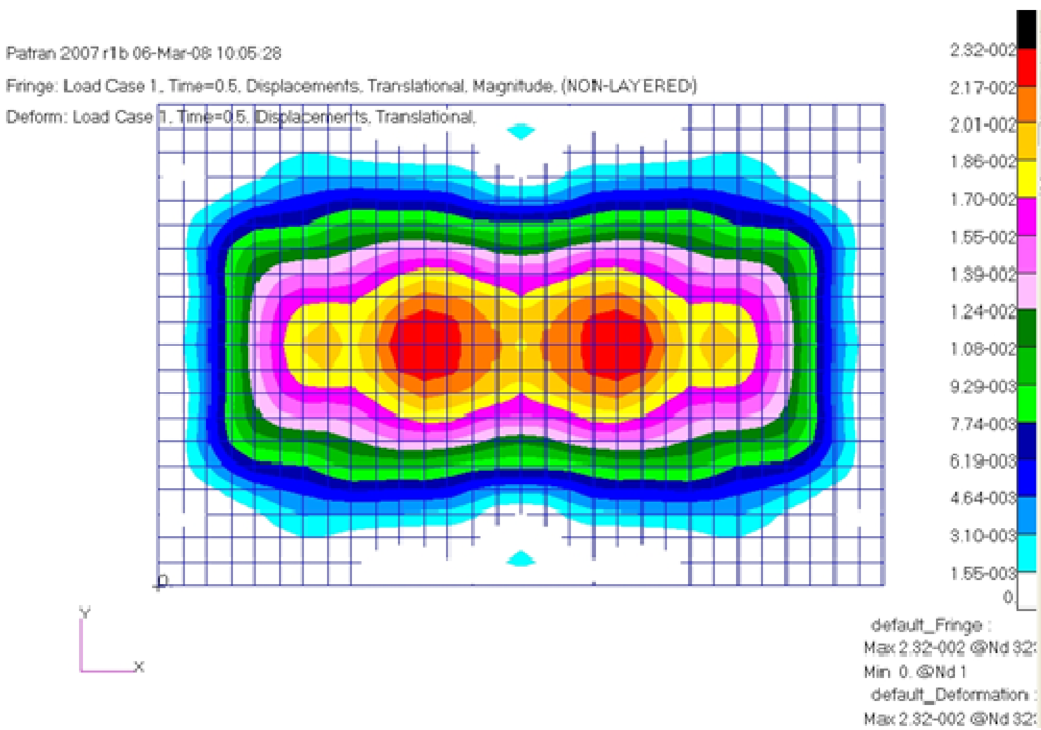

Figure 26. Displacements across an 8 cm thick steel plate.

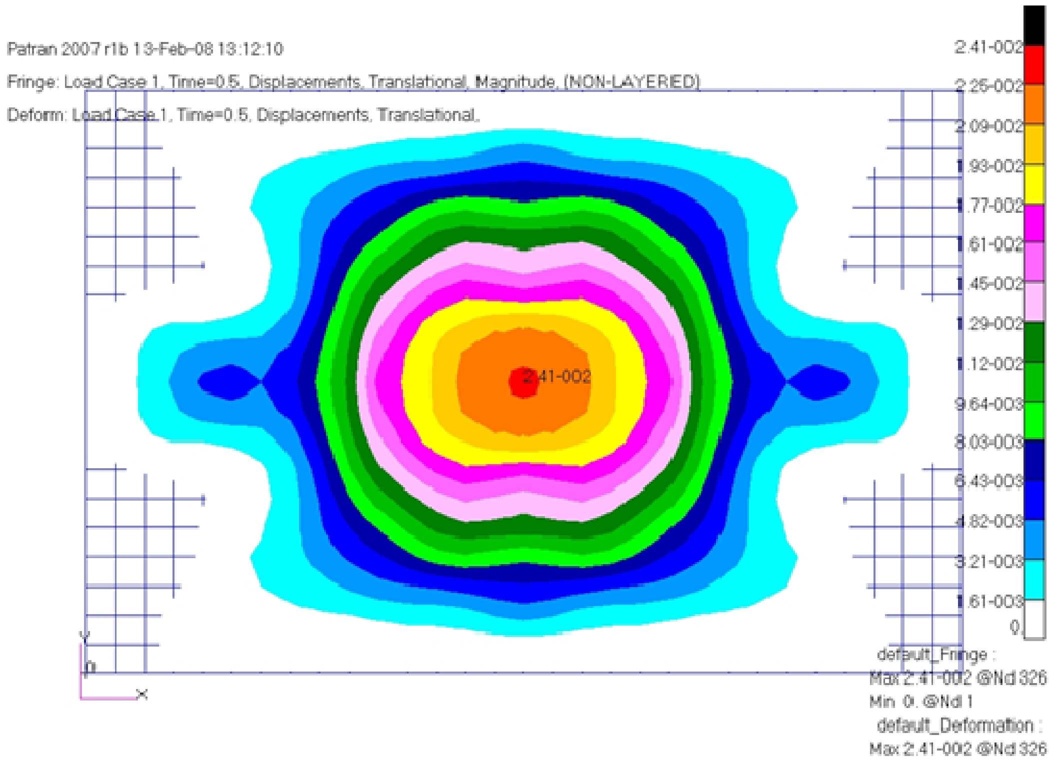

Figure 27. Displacements across a 10 cm thick steel plate.

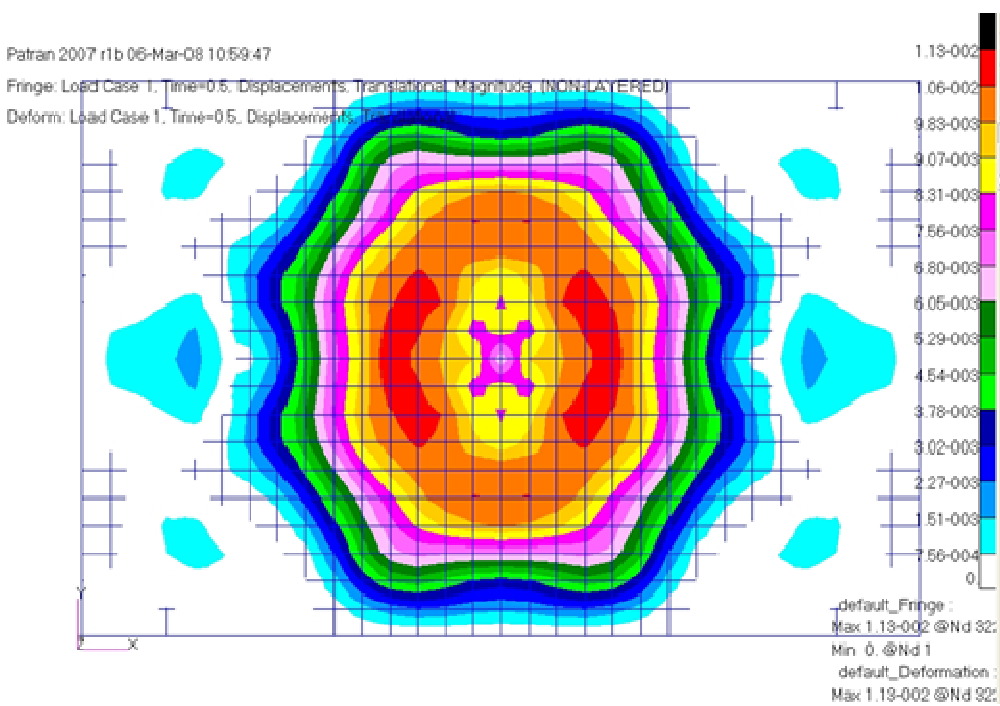

Figure 28. Displacements across a 12 cm thick steel plate.

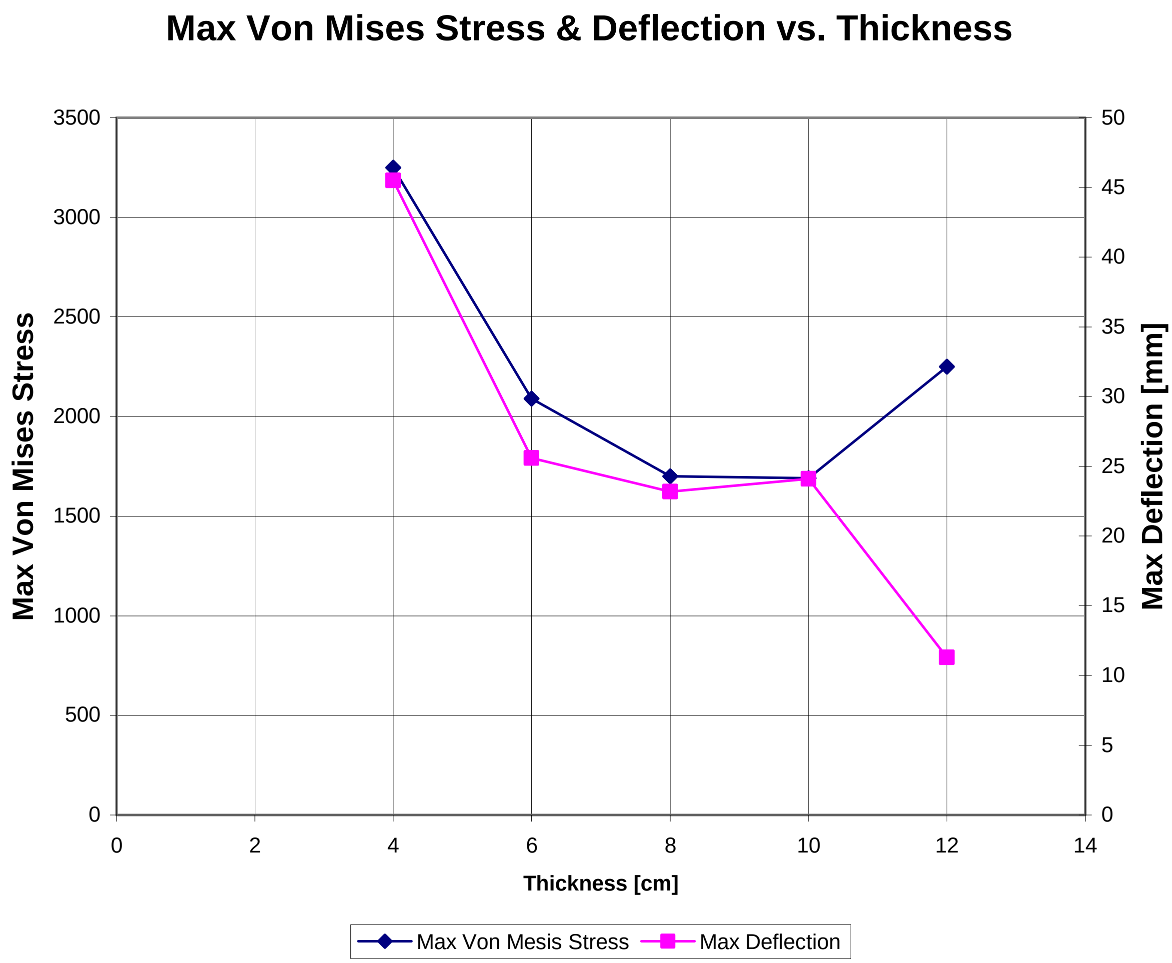

Figure 29. Chart of steel's performance at different thicknesses.

Figure 29 shows the effect of thickness on the maximum deflection & Von Mises stress for the armor plate. It is observed that as the thickness increases the maximum stress moves from the center to the edges. The maximum stress also decreases as thickness increases from 4cm to 10 cm followed by an increase at 12 cm. In all but the 4 and 8 cm thick plates the maximum Von Mises stress occurs at the top and bottom constraint, while in the 8 cm thick plate the maximum Von Mises stress occurs on the left and right side. In the 4 cm thick plate the maximum Von Mises stress occurs between the center and the top and bottom constraints. The discrepancies in the location of the

maximum Von Mises stress may be attributed to the short simulation run time and/or the whipping of the armor plate as a result of explosive shock. The increase in the maximum stress at 12 cm of thickness may indicate the limitations of 2D plate analysis, where the thickness can violate the assumptions on the lateral normal stress and transverse shear stresses.

B. COMPOSITES WITH INTER-LAMINAR SHEAR

This section presents analysis results for laminated composite design using TSAI-WU criteria. Figure 30 and 32 present the stresses and strains for the mechanical loads only.

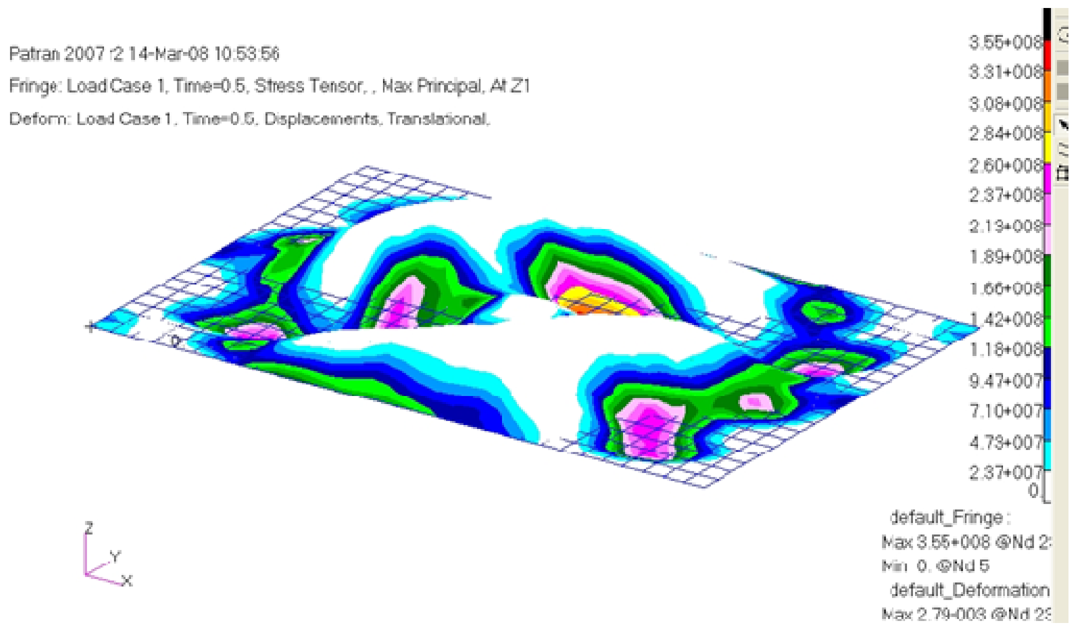

Figure 30. Tsai-Wu criteria stresses in high-modulus graphite.

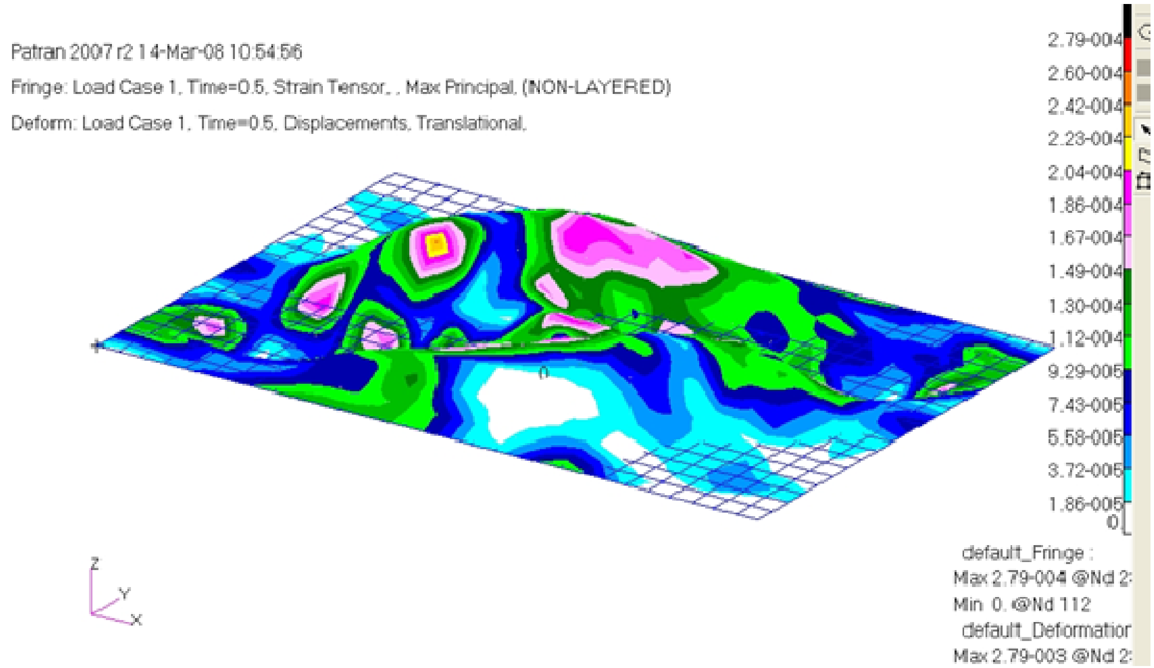

Figure 31. Tsai-Wu criteria strains in high-modulus graphite.

When Figure 31 is compared to Figures 11 and 12, high-modulus and high-strength graphite as an orthotropic material, the resulting stress is reduced to about 50% of yield strength while the graphite material had failed. Another significant change in the results is that the stress and strain is distributed throughout the entire plate, taking maximum advantage of composites unique tailoring properties compared to metals. The resulting strains, seen in Figure 31, are safely below failure criteria for typical composites. This advantage may decrease with decreases in the size of the plate.

C. THERMAL LOADING RESULTS

The results from the thermal analysis are presented in the following section. The thermal loads applied to the armor plate include the ambient temperature and the initial temperature of the EFP.

Figure 32. Temperature distribution on an aluminum plate.

Figure 33. Temperature distribution on a steel plate.

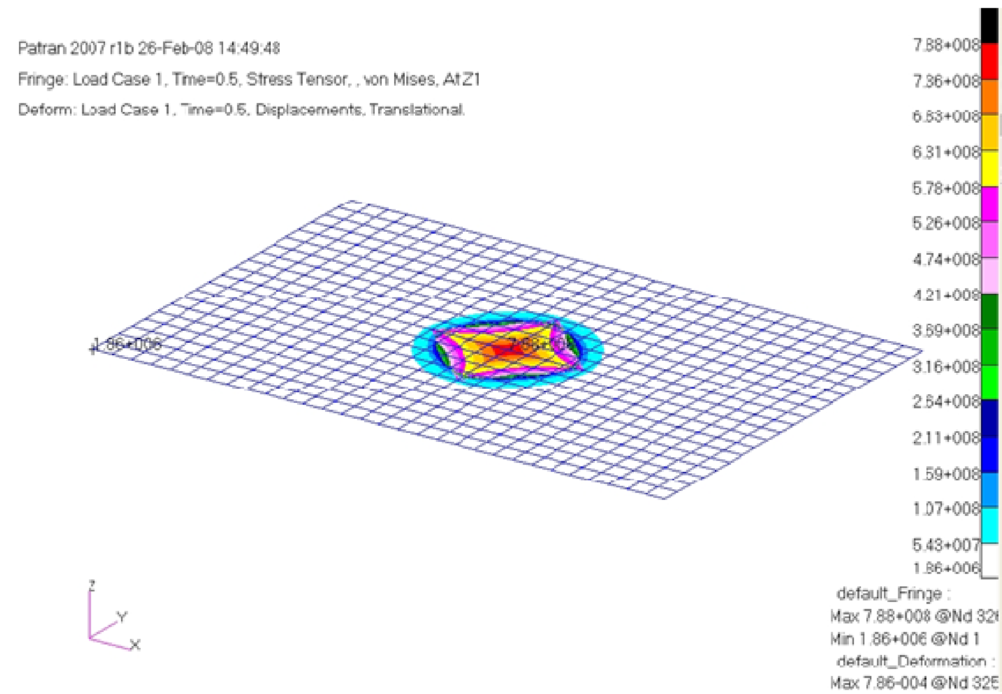

Figure 34. Von Mises stress distribution caused by initial temperature on aluminum.

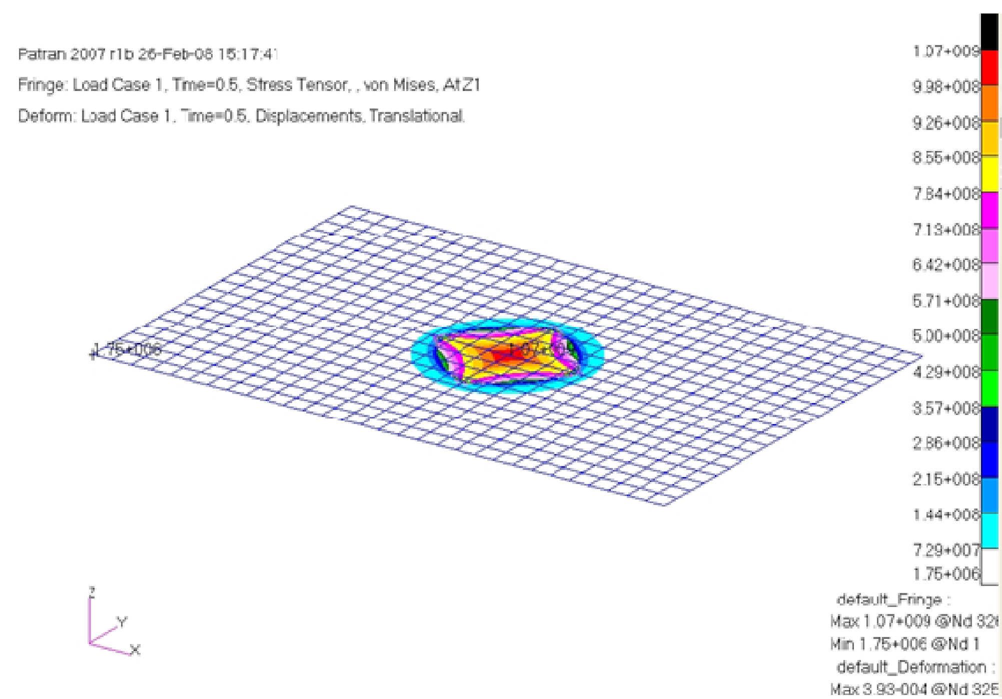

Figure 35. Von Mises stress distribution caused by initial temperature on steel.

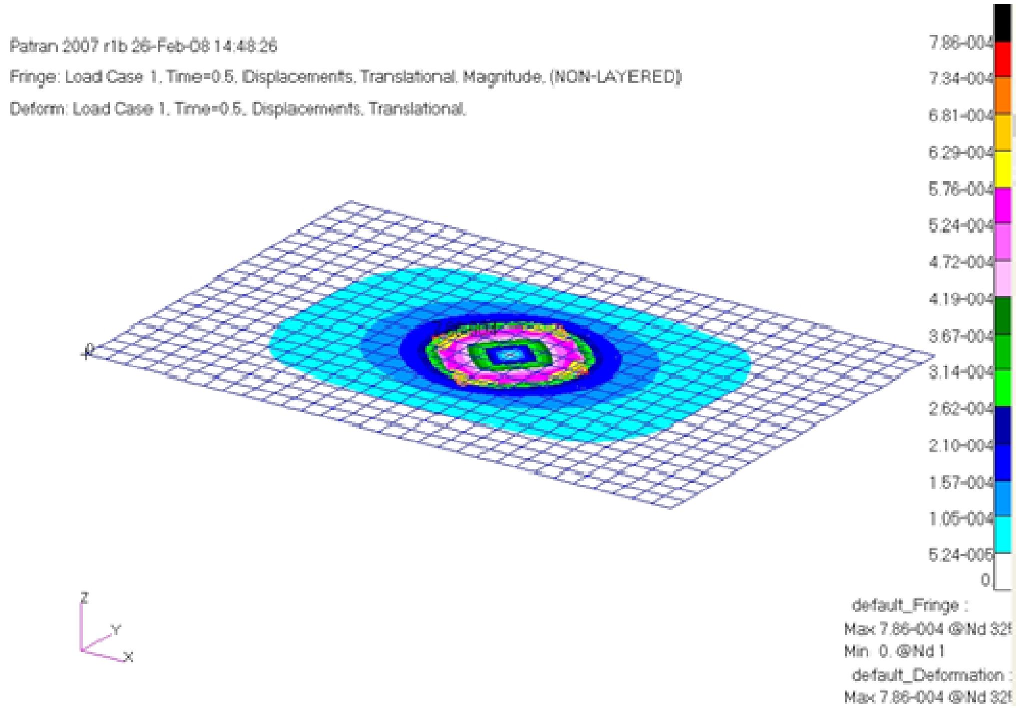

Figure 36. Displacements from initial temperature on aluminum.

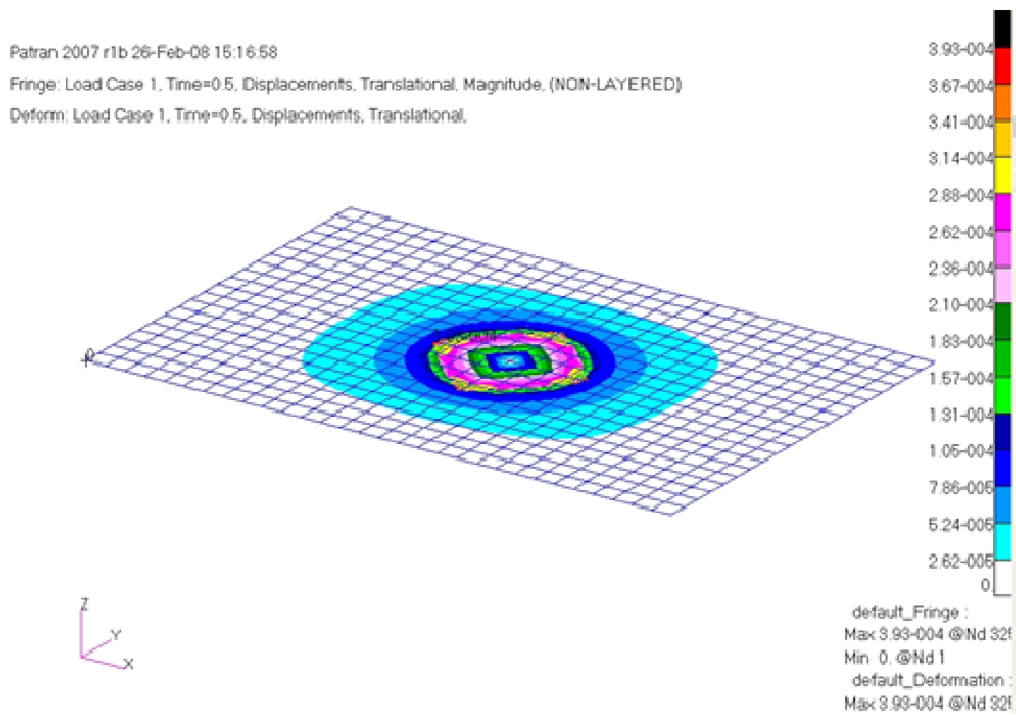

Figure 37. Displacements from initial temperature on steel.

The results show that the heat from the EFP does not dissipate very much in one half second of contact. The stresses caused by the thermal loading are of the same order of magnitude as the stresses caused by the kinetic loads. The resulting deflections are relatively small compared to those caused by the kinetic loads.

D. 3-D THERMAL LOADING RESULTS

The three dimensional plate was composed of ten nodded tetrahedrons increasing the accuracy of the analysis by having higher order approximations for the unknown variables (temperatures and displacements).

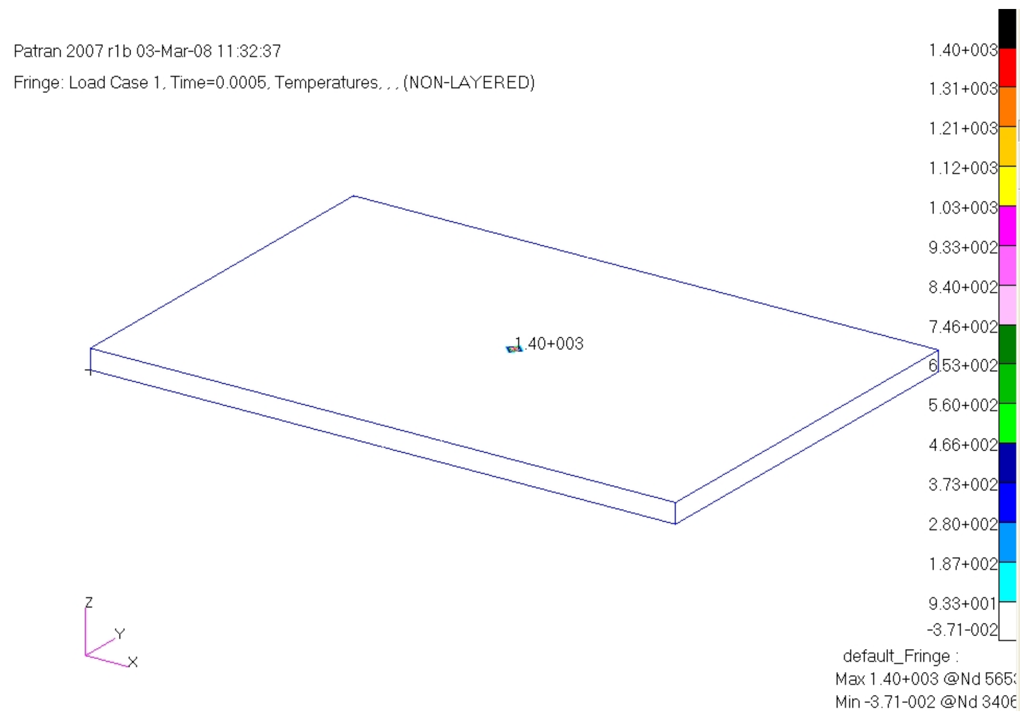

Figure 38. Initial temperature acting on a 3d aluminum plate at t = 0.0005.

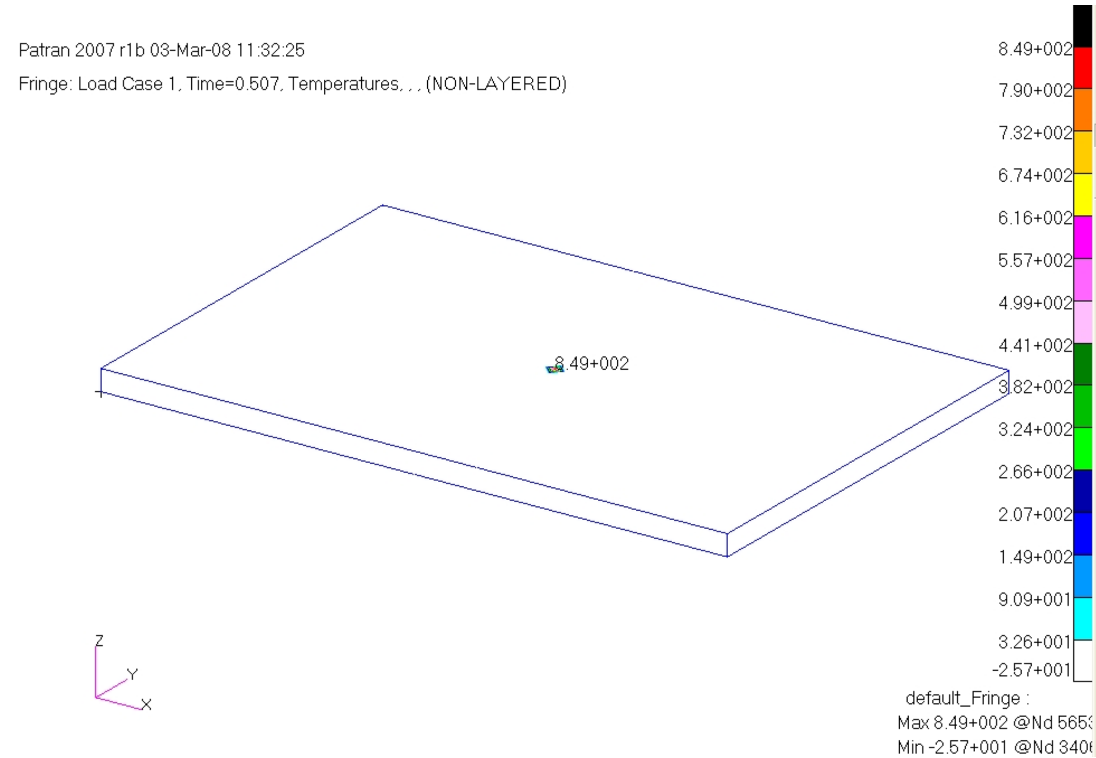

Figure 39. Initial temperature acting on a 3d aluminum plate at t = 0.507.

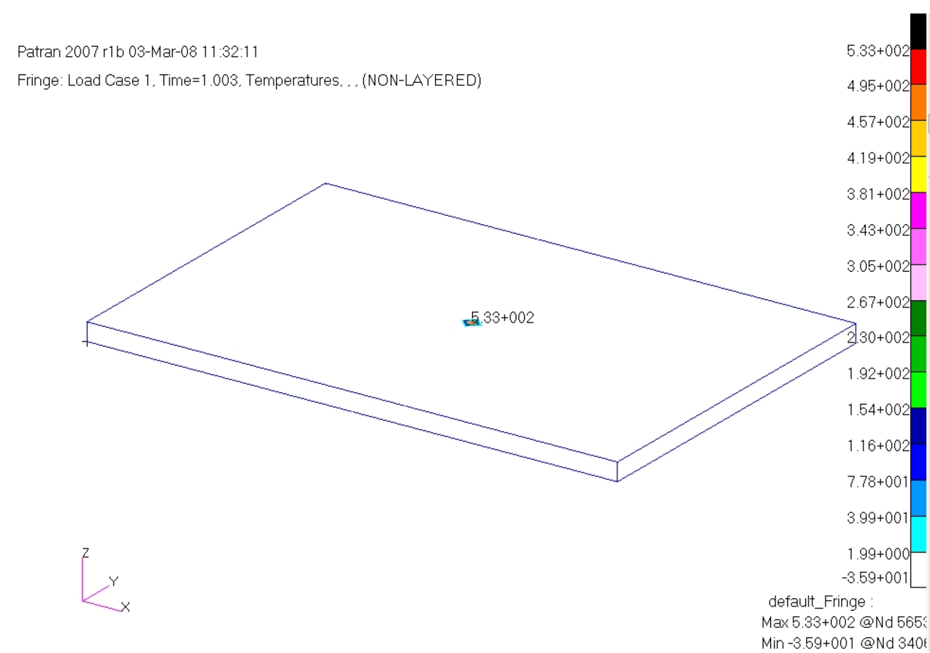

Figure 40. Initial temperature acting on a 3d aluminum plate at t = 1.003.

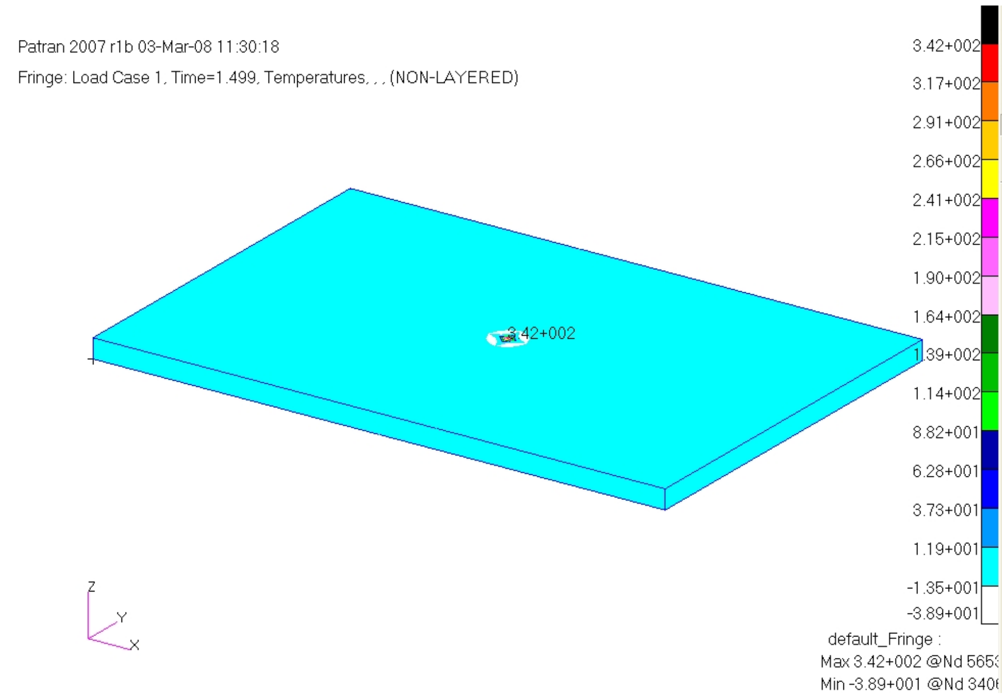

Figure 41. Initial temperature acting on a 3d aluminum plate at t = 1.499.

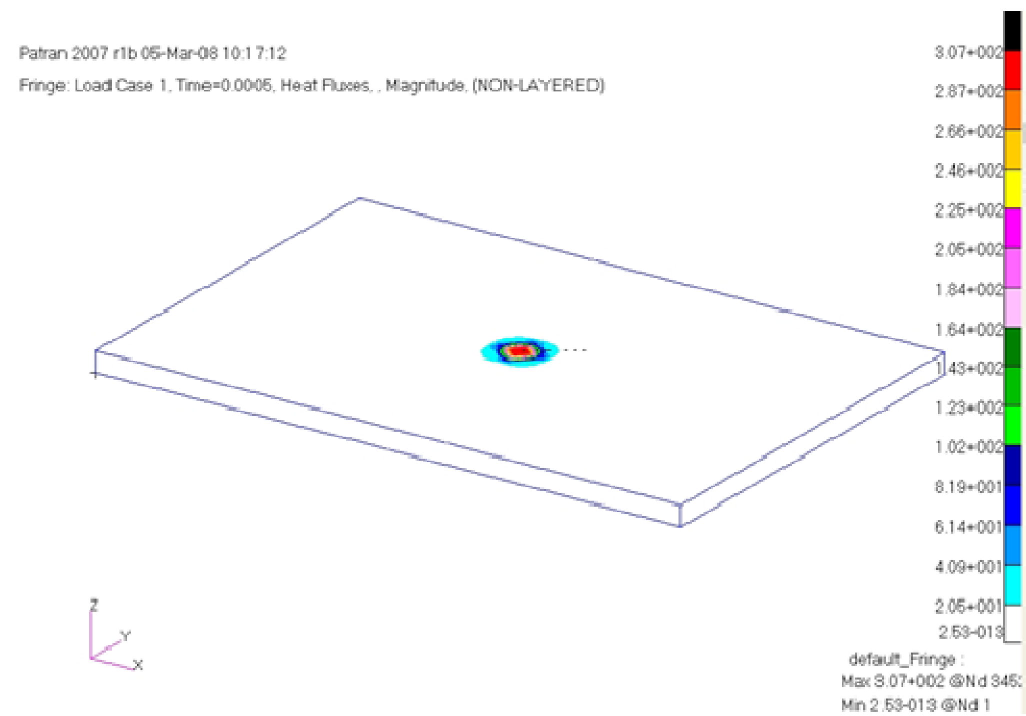

Figure 42. Heat flux acting on a 3d aluminum plate at t = 0.0005.

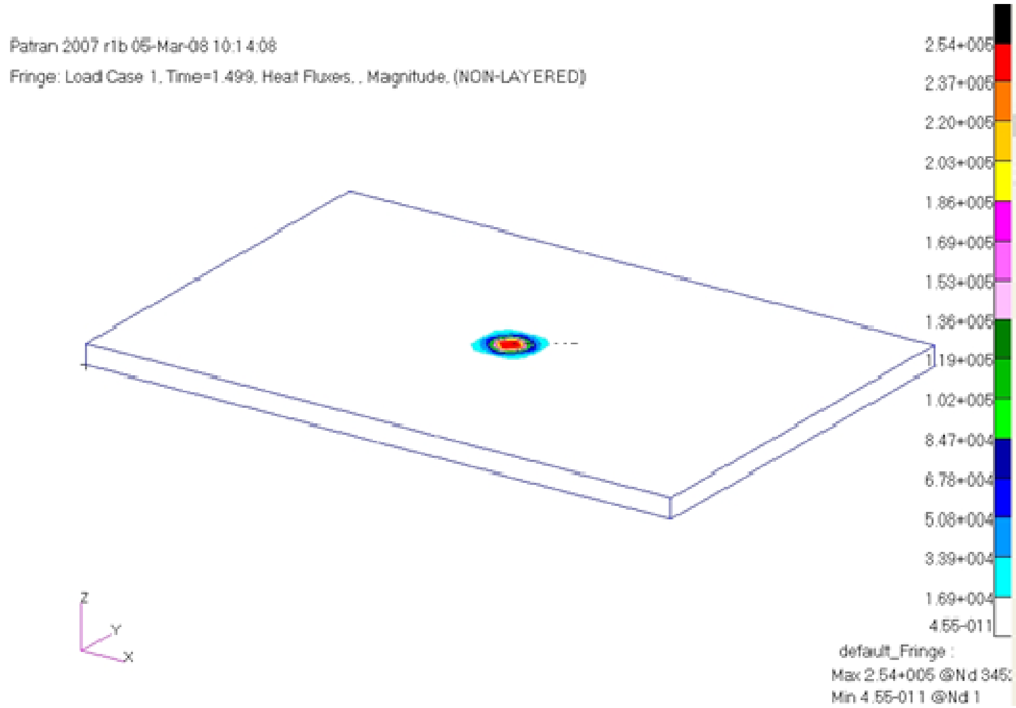

Figure 43. Heat flux acting on a 3d aluminum plate at t = 1.499.

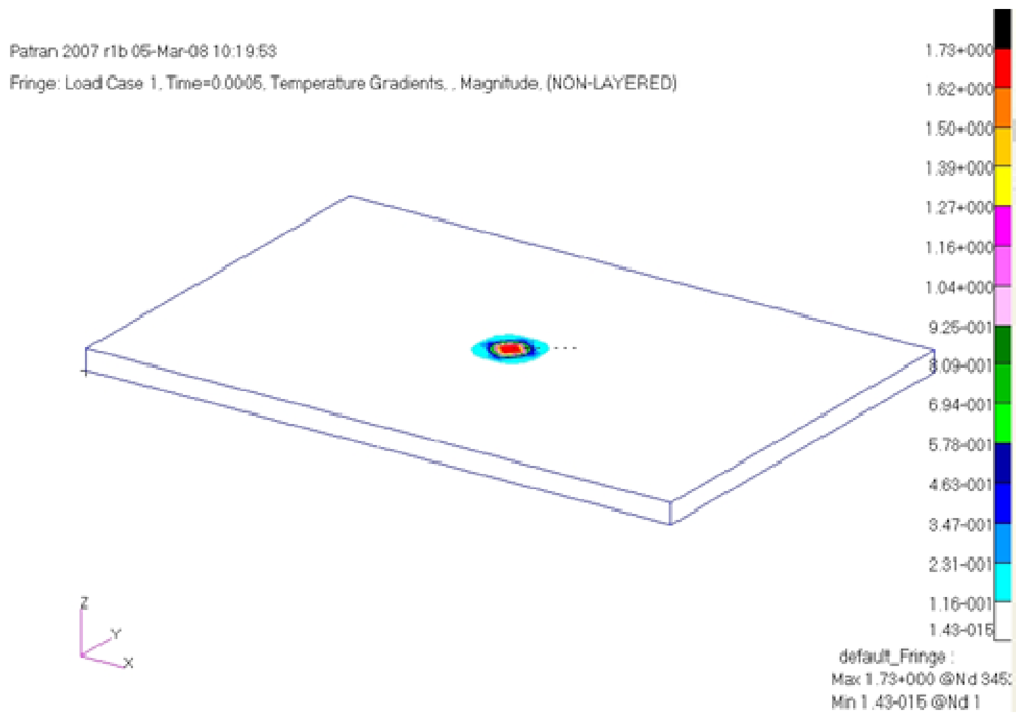

Figure 44. Temperature gradient across a 3d aluminum plate at t = 0.0005.

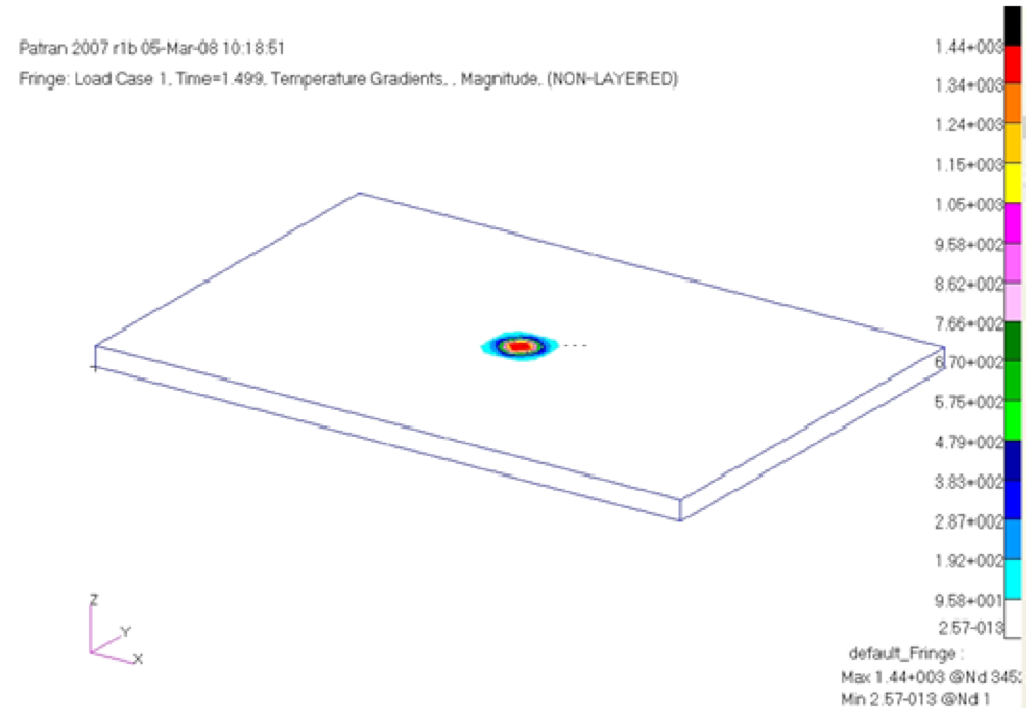

Figure 45. Temperature gradient across a 3d aluminum plate at t = 1.499.

As shown in Figures 38 through 41 the initial temperature drops from a high of 1400 K to a low of 342 K on the surface in 1.5 seconds while raising the rest of the aluminum armor plates temperature by 11.9 K. The EFP's point of impact has to absorb large amounts of energy as shown in Figures 42 and 43. Figures 44 and 45 show how the temperature gradient at the point of impact increases rapidly from 173 k/m to 1440 k/m.

E. 3-D COMBINED LOADING OF ALUMINUM

A course mesh was selected for the 3-D analysis resulting in 2,000 DOF (degrees of freedom) as compared to 20,000 DOF. The high fidelity model needs enormous computing resources for these simulations and was not pursued. The following results are for a 10 cm thick plate of aluminum armor plate loaded simultaneously with the pressure wave, initial velocity, and the thermal loading.

Figure 46. Temperature gradient across a 3d aluminum plate at t = 0.5 sec.

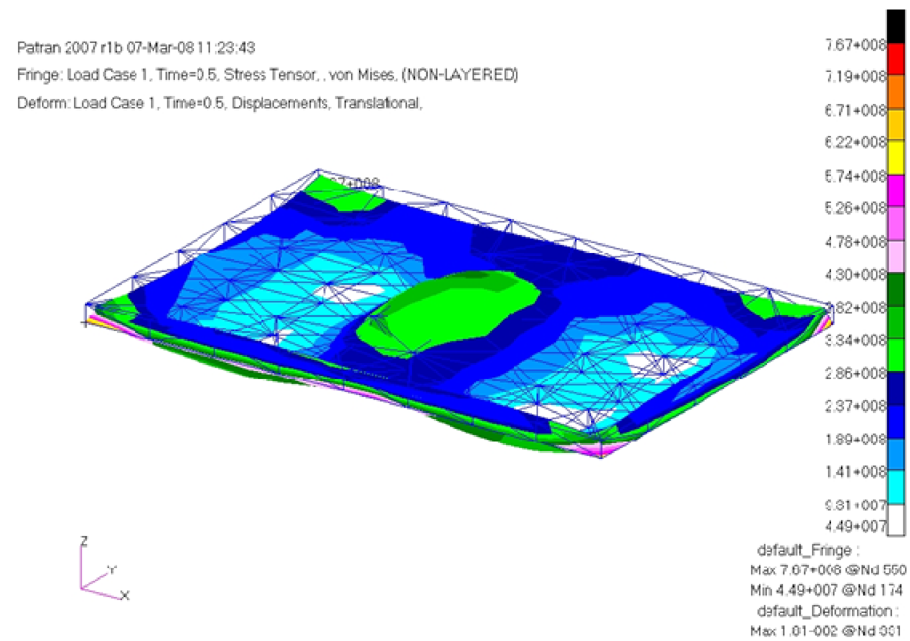

Figure 47. Maximum von Mises stress on a 3d aluminum plate at t = 0.5 sec.

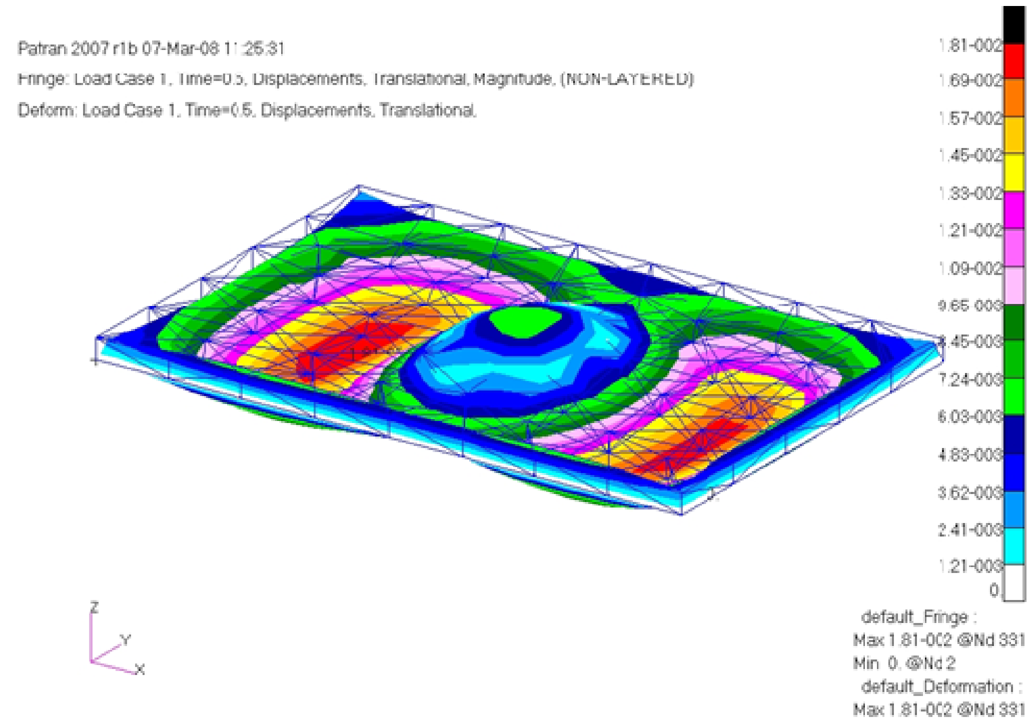

Figure 48. Maximum displacements on a 3d aluminum plate at t = 0.5 sec.

As shown in Figures 47 and 48 the maximum stresses and displacements are similar to the 2D analysis shown in figures 7 and 13. In both cases the metal will yield and require an elastic-plastic analysis to determine any further failure information. The 3D analysis does show a more realistic model of the distortions caused by the impact of the EFP.

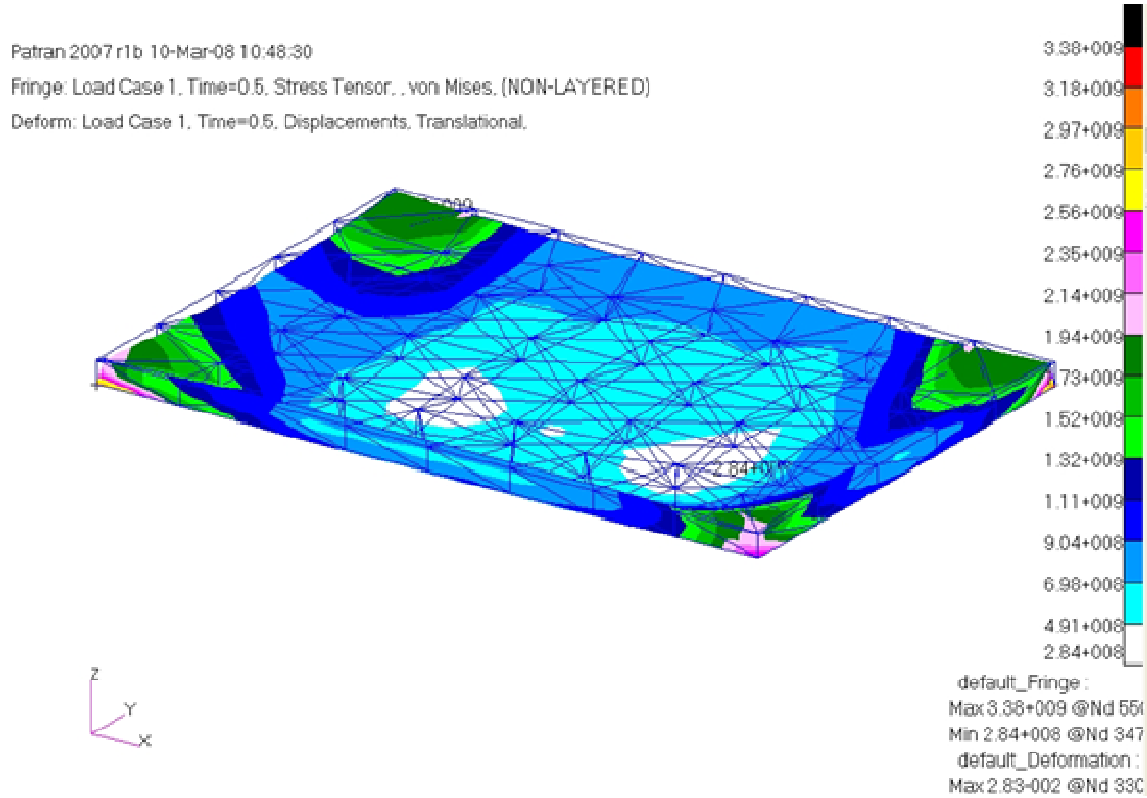

Figure 49. Maximum von Mises stress on a 3d steel plate at t = 0.5 sec.

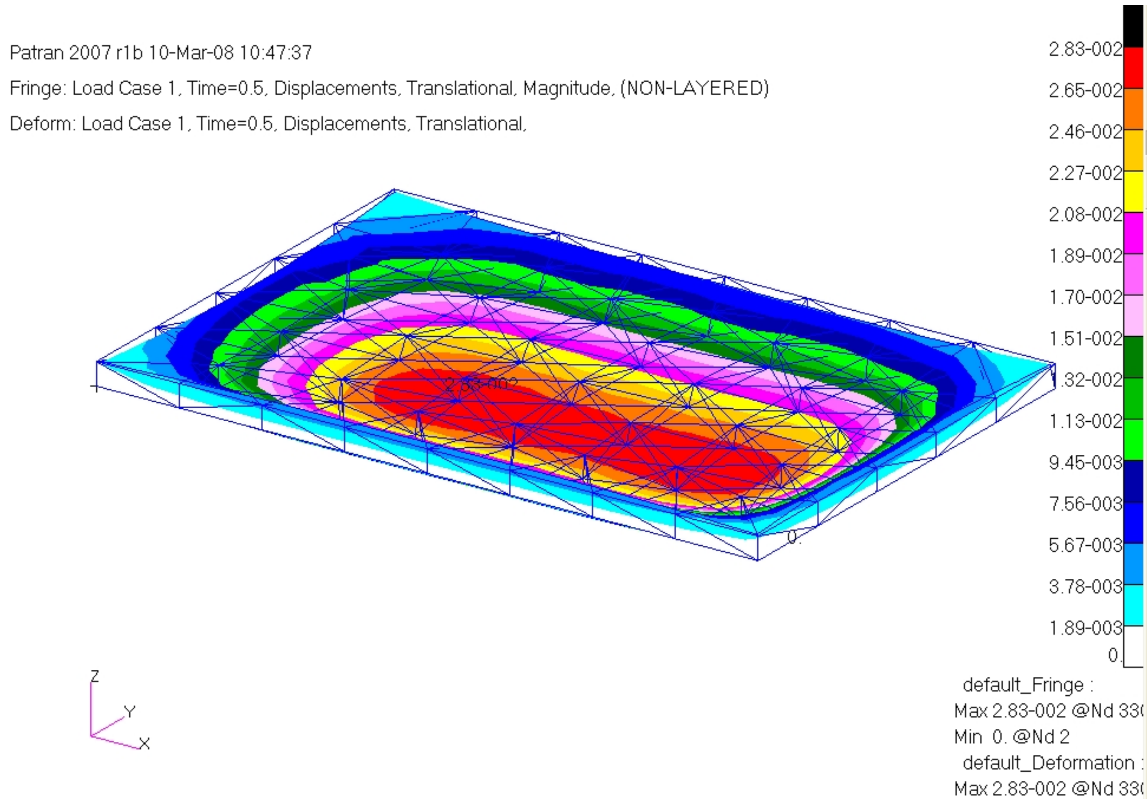

Figure 50. Maximum displacements on a 3d steel plate at t = 0.5 sec.

The maximum Von Mises stress in the two dimensional model is an order of magnitude larger than the maximum stress in the three dimensional model as shown in Figures 8 and 49. The corresponding maximum deflections are 24 mm and 29 mm respectively as shown in Figures 14 and 50

THIS PAGE INTENTIONALLY LEFT BLANK

VI. CONCLUSION

A methodology was developed and demonstrated to model and simulate survivable armor plates subjected to specified requirements. The design constraints included initial temperature flux doe to EFP, initial velocity due to EFP, and a mathematical representation of blast loading.

The parametric design study included variations of different materials, different thicknesses, and configurations that included composite construction.

The studies indicate that modeling the armor plate with 2-D plate elements are inadequate due to thickness effects and 3-D elements capture the behavior of armor. This can be evidenced in Table 1 and Figure 29.

Thickness [cm]	Max Stress Von Mises [MPa]	Max Deflection [mm]	Weight [kg]
4	3250	45.5	1,886
6	2090	25.6	2,830
8	1700	23.2	3,773
10	1690	24.1	4,716
12	2250	11.3	5,659

Table 1. Summary of key data for steel of various thickness x 3 m x 2 m.

The high-strength graphite composite performed the best followed by the high-modulus graphite composite when the weight is considered as show in Table 2. The analyzing software however did not consider the thermal stresses in composites. A recommendation would be to use a sandwich construct of aluminum/high-strength

composite with aluminum sheet exposed to the high temperature/velocity impact, this approach would take advantage of both the material's properties for the armor design.

	Density [kg/m^3]	Weight [kg]
Steel	7,860	4,716
Aluminum	2,800	1,680
High-Mod Graphite	1,200	720
High-Str Graphite	1,400	840

Table 2. Weight of 3 m x 2 m x 0.1 m armor plate made of listed materials.

Follow up analysis with this EFP model could incorporate ballistic ceramic materials. The methodology established could be used to incorporate candidate materials and field relevant blast loads and initial velocity estimates to design the armor to survive and be less vulnerable to the IED threat.

Any armor successful at defending against the insurgents IED attacks will result in the insurgents using bigger IEDs. Even the composite materials used here were heavy and may prove impractical on military vehicles. In an overall survivability approach it may prove more cost effective to reduce the susceptibility of military vehicles, with better sensors and countermeasures, vice building an oversized vehicle with decreased vulnerability.

APPENDIX

A. ARMOR PLATE CONSTRUCTION

This section will discuss how to create the armor plate in PATRAN starting with opening a new file, and ending with applying the boundary conditions.

1. Our First Step is to Create a New Data Base

From the top menu choose File, and in the resulting pull down menu choose New. A side menu called New Database pops up:

- Turn on (check) Modify Preferences.
- Under Look In select the computer directory you want to work in.
- Under File Name enter the name you want to give the file.

2. Next Set the Analysis Preference

A New Model Preferences will appear as a side menu on the right hand side:

- Under Tolerance choose based on model.
- Set Dimensions to 10.0.
- Under Analysis Code choose MSC.Nastran
- Under Analysis Type choose Structural.

3. The Geometry of the Beam will Beam Determined Next

From the top menu choose the Geometry button. A side menu called geometry will open on the right hand side:

- Under Action select create.
- Under Object select solid.

- Under Method select XYZ
- In the Vector Coordinates List box enter the dimension of the armor plate. For example <3, 2, 0.1>.
- Uncheck Auto Execute.
- In the Origin Coordinates List enter the point you want the armor plate to originate at. Only the z coordinate should change base the how layers the plate is made of.
- Click Apply
- Repeat the geometry steps as needed to create multiple layers. Adjusting the Origin Coordinates List as needed.

4. Specify the Boundary Conditions Next

From the top menu choose the Loads/BCs button. A side menu called Load/Boundary Conditions will appear on the right hand side:

- Under Action select create.
- Under Object select displacement.
- Under Type select nodal.
- Set Current Load Case to default.
- In the New Set Name box enter the name you want to give the boundary condition.
- Click the Input Data button, now enter <0,0,0> in the Translations box and in the Rotations box, and click ok.
- Click the Select Application Region button, on the mouse pointer select menu depress the Curve or Edge button. Now select the edges to be clamped so there label appears in Select Geometry Entities box and click add. When finished adding click ok.
- Click Apply.

5. The Finite Element Mesh is Specified Next

From the top menu choose the Elements button. A side menu will appear on the right had side.

- Under Action select create.
- Under Object select mesh.
- Under Type select solid.
- Under Elem Shape select tet.
- Under Mesher select IsoMesh.
- Under Topology select Tet10.
- In the Solid List box enter all solids by highlighting them with the mouse.
- In the Global Edge Length Box make sure Automatic Calculation is checked.
- Click Apply

If using multiple layers perform the following.

- Under Action select equivalence.
- Under Object select all.
- Under Method select tolerance cube.
- In the Equivalencing Tolerance box enter 0.005.
- Click Apply

B. ARMOR PLATE LOAD CASE CONSTRUCTION

1. The Load Case is Created Next

From the top menu choose Load Cases. A side menu will appear on the right hand side.

- Under Action select create.
- In the Load Case Name box enter the name of the load case for example “LC1”.
- Click apply.

2. The Loads are Specified Next

From the top menu choose the Loads/BCs button. A side menu called Load/Boundary Conditions will appear on the right hand side:

- Under Action select create.
- Under Object select pressure.
- Under Type select element uniform.
- In the Current Load Case box click the button and select the load case you created.
- In the New Set Name box enter the name of the pressure force.
- Click the Input Data button and enter,
  i. Translations <0,0,45000>.
  ii. Rotations <0,0,0>.
  iii. Click ok.
- Click Select Application Region,

i. In the Application Region box enter the name of the surface the pressure is being applied to by highlighting it in the work window with the mouse pointer set to Surface or Face.

ii. Click add, the name of the surface will appear in the Application Region box.

iii. Click ok.

- Click apply.

The initial velocity is created next.

- Under Action select create.
- Under Object select initial velocity.
- Under Type select nodal.
- In the Current Load Case box click the button and select the load case you created.
- In the New Set Name box enter the name of the initial velocity.
- Click the Input Data button and enter,

i. Translations Veloc <0,0,-2000>.

ii. Rot Veloc <0,0,0>.

iii. Click ok.

- Click Select Application Region,

i. In the Application Region box enter the name of the node the initial velocity is being applied to by highlighting it in the work window with the mouse pointer set to Point or Vertex.

ii. Click add, the name of the node will appear in the Application Region box.

iii. Click ok.

- Click apply.

C. CREATING ARMOR MATERIAL

1. The Materials are Specified Next

From the top menu choose the Materials button. A side menu called Materials will appear on the right hand side:

- Under Action select create.
- Under Object select isotropic for metals and 2d orthotropic for composites.
- Under Method select Manual input.
- In the Material Name box enter the name of the material (i. e. Steel).
- Click the Input Properties button for,
    i. Under Constitutive Model select Linear Elastic. For metals enter the values of Elastic Modulus, Poisson Ratio, Shear Modulus, and Density. For composites enter the values of Elastic Modulus 11, Elastic Modulus 22, Poisson Ration 12, Shear Modulus 12, Sheer Modulus 23, and Density.
    ii. Under Constitutive Model select Failure. For metals enter the values of Tension Stress Limit, Compression Stress Limit, and Shear Stress Limit. For composites select Failure1 and enter the values of Tension Stress Limit 11, Tension Stress Limit 22, Compress Stress Limit 11, Compress Stress Limit 22, Shear Stress Limit.
    iii. Click ok.
- Click apply.

Material	High Strengt ASTM-A242	Aluminum Alloy 2014-T6	High-Modulus Graphite Epoxy (GY-70-HYE1534)	High-Strength Graphite_Epoxy (AS-3501)
Ultimate Strength [Mpa]				
Tension	480	480		
›mpression	480	480		
Shear	480	290		
Modulus of Elasticty [Gpa]				
E1	200	72	290	128
E2			6.9	11
Modulus of Rigidity [Gpa]				
G	79	28	4.8	4.5
Piosions's Ratio				
Nu	0.30	0.30	0.25	0.25
Density [kg/m3]				
Ro	7860	2800	1200	1400
Yield Strength [Mpa]				
Tension	345	410		
Xt			621	1170
Xc			621	1120
Yt			13.8	41
Yc			193	172
Shear	210	220		
S			27.6	48
Coefficient of Thermal Expansion [10^-6 C]				
alpha1	11.7	23	-0.32	0.14
alpha2			9.17	8.44
Specific Heat [J/kg * K]				
Cp	446	875	935	935
Thermal Conductivity [W/m * K]				
k parallel	51.9	177	11.1	11.1
k perpendicular			0.87	0.87

Table 3. Properties of various tested materials.

Creating a Composite

While, still working under the materials menu:

- Under action select create.
- Under object select composite, a Laminated Composites window will appear.
- In the insert box enter the number of layers to be created.
- For each row click the box under the material heading and select a material from the materials list showing on the materials right hand menu.
- Next click in the box under the thickness heading enter the thickness in the in the input data box.
- Do the same for the box under the orientation heading.
- When done entering laminate layers click the click the show laminate properties button, and the properties of the laminate will be displayed in matrix form.
- To close the laminate select another option under object or choose another top menu option.

2. The Properties are Assigned Next

From the top menu choose the Properties button. A side menu called Properties will appear on the right hand side:

- Under action select create.
- Under object select 2D.
- Under type select shell.
- In the Property Set Name box enter a name.
- Click the Input Properties button, an input properties window will apper.

i. Click the cross hatch icon next to Mat Prop Name and select a material from the list that appears.

ii. For 2D objects enter the thickness in the thickness box

iii. Click ok.

- Click the Select Application Region button, a new menu will open on the right.

i. Click in the Select Members box, then go to the work space and select the whole armor plate.

ii. Click the add button, the name of the whole plate will appear in the Application Region box.

iii. Click ok.

- Click Apply

D. KINEMATIC LOAD ANALYSIS

1. Analysis is Performed Next

From the top menu choose the Analysis button. A side menu called Analysis will appear on the right hand side:

- Under action select analyze.
- Under object select entire model.
- Under method select analysis deck.
- In the job name box enter a job name.
- Click the Translation Parameters button.

i. Select OP2, and click ok.

- Click the Solution Type button.

i. Select Transient Response for the solution type.

ii. Under Formulation select direct.

iii. Click ok.

- Click the subcases button.

i. In the Available Subcases box highlight the subcase to be analyzed, the name will appear in the Subcase Name box.

ii. Highlight the load case to be analyzed in the Available Load Cases box.

iii. Click the Subcase Parameters button, then click the Define Time Steps button.

iv. Enter the number of time steps and the value of delta t, then click ok.

v. Click ok on the Subcase Parameters box.

vi. Click the Out Put Requests box, and select the desired outputs form the menu, and click ok.

vii. Click Apply in the Subcases window, then cancel to close the window.

- Click the subcase select button.

i. Highlight the desired load case in Subcases For Solution Sequence box.

ii. Remove any extra load cases from the Subcases Selected box.

iii. Clock ok.

- Click Apply, PATRAN will create the .bdf file in temp directory.
- Next open Nastran, select the .bdf file and click open.
- In the option keywords box enter, “news=no,” and , “scr=yes”.
- Click Run.

2. Accesses the Results

Open a database file. Then click the Analysis button.

- Under action select access results.
- Under object select read output2.
- Under method select both.
- Click apply.
- Then from the top menu click Results.
- View the results creating images or videos of desired data.

E. THERMAL LOAD ANALYSIS

From the top menu choose the Loads/BCs button. A side menu called Load/Boundary Conditions will appear on the right hand side:

- Under Action select create.
- Under Object select initial temperature.
- Under Type select element uniform.

F. THERMAL STRESS ANALYSIS

To perform a stress analysis you will need open a new data base, and click the Analysis button on the top menu.

1. Accesses Thermal the Results

- Under action select access results.
- Under object select read output2.
- Under method select both.

- Click Results on the top menu, and select any result to view, you must do this for a later state.

2. Create a Spatial FEM Based on the Temperature Profile

- Under action select create.
- Under object select spatial.
- Under method select FEM.
- In the Field Name box enter a name.
- In FEM file definition select continuous.
- In Mesh/Results group filter select Scalar.
- In select group select default group, if you did choose a set of result this will be blank.
- Click apply

3. Change the Analysis Type

- Select Preferences/Analysis for the menu above the top menu.
- Select Analysis type, choose structural, and click ok.
- Add or change the materials as before if needed.
- Add or change the properties as before if needed.
- Create a new load case as before, the constraint forces will need to be added or the temperature distribution can be added to an existing load case.

4. Define a Temperature Load

- Under action select create.

- Under object select temperature.
- Under type select Nodal.
- In the New Set Name box enter the name of the temperature load.
- Click the input data button, and select the name to the temperature from the thermal analysis, and click ok.
- Click the select application region button, choose the whole plate of armor, and click ok.
- Click Apply
- Perform the analysis as before.

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF REFERENCES

[1] B. Vanderstraeten, M. Lefebrev, J. Berghmans, "A Simple Blast Wave Model for Bursting Spheres Based on Numerical Simulation", Journal of Hazardous Materials, vol. 46, pp. 145-157, 1996.

[2] Ansel C. Ugural and Saul K. Fenster, Advanced Strength and Applied Elasticity. Prentice Hall, 2003.

[3] William T. Thomas and Marie Dillon Dahlen, Theory of Vibration with Applications. Prentice Hall, 1998.

[4] Frank P. Incorpera and David P. DeWitt, Introduction to Heat Transfer. John Wiley & Sons, 1996.

[5] Robert M. Jones, Mechanics of Composite Materials. Edwards Brothers, 1999.

[6] Robert E. Ball, The Fundamentals of Aircraft Combat Survivability Analysis and Design. American institute of Aeronautics and Astronautics, Inc., 2003.

[7] David Crane, "New Vehicle Armor Developed to Defeat Explosively Formed Penetrators (EFPs)"January 2008, http://www.defensereview.com/modules.php?name=News&file=article&sid=1026.

[8] Sang H. Lee, Editor, MSC/NASTRAN Nonlinear Analysis Handbook. The Macneal-Schwendler Corporation, 1992.

THIS PAGE INTENTIONALLY LEFT BLANK

INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center
   Ft. Belvoir, Virginia

2. Dudley Knox Library
   Naval Postgraduate School
   Monterey, California